Introduction to Drafting and Autodesk Inventor

Introduction to Drafting and Autodesk Inventor

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Part 5

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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
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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.
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This book was produced using the following styles: Autodesk Inventor Style Sheet [Word file]

Introduction

Learning Outcomes

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

1. List the software and template files required to complete the Autodesk Inventor book.
2. Describe the Autodesk Inventor book's philosophy and explain how to use the book to learn how to apply the Inventor commands and features to create Inventor parts, assemblies, and drawings.

Autodesk Inventor Software Required

Introduction to Drafting and Autodesk Inventor was written to be used with Autodesk Inventor Version 2020 and 2021. You must have access to one of these software packages to complete the lab exercises in this book. It can also be used with Inventor 2015 through to 2019, but the interface structure was different and will take a little more poking around to find things.

About the Book

Philosophy of the Inventor book

This book contains self-paced learning modules that were written as a tool to guide and teach you to master Inventor. No two students learn at the same pace, therefore, the modules were written as competency-based bite-size pieces to allow you to work at your own pace. They can be used in correspondence courses, online courses, instructor-lead classes or by individuals teaching themselves to use Inventor in their own home or office.

Scope of the Inventor books

The Inventor books were written in two parts, Autodesk Inventor and Autodesk Inventor Advanced.

Suggested Prerequisites

To get the most from this book, it is suggested that you have a working knowledge of the Windows operating system that is installed on the computer you are using to learn Inventor. You must be able to send and receive e-mails, complete with attachments. You must also be able to create folders, save files, move files, copy files, rename files, and delete files.

Book Structure and Components

The Autodesk Inventor book modules were written in a very logical step-by-step order. To get the most from this book:

Do not skim through them.

You must read and understand everything in each module.

Do not jump around inside the module or from module to module. Work your way methodically through each module, page by page.

The Modules

The modules were written in a precise order and contain the information you require to learn Inventor. Using Inventor is a never ending learning process and you will continue learning long after you complete these modules.

As you work your way through each module, learn and try to understand all of the material. To ensure that you understand it, complete the lab exercise(s) to prove it. Only when you understand everything in the module and can complete the lab exercise(s) in the specified time limit, should you go to the next module.

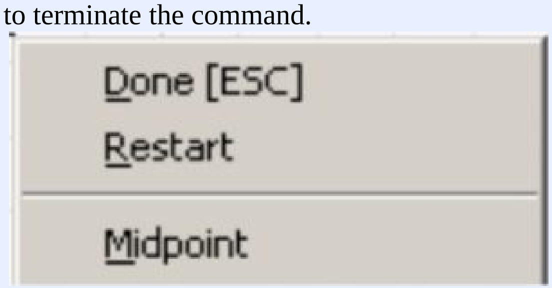
Each module may contain Must Know's, User Tips, Workalongs, Drafting Lessons, Geometry Lessons, and Lab Exercises.

Must Knows

A *Must Know* is a an important Inventor principle or fact. 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, get some help or read back through the module. Do not go on until you fully understand it. See the example below:

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

MUST KNOW: The ESC key or the right-click menu can be used to end the current command. You must let Inventor know when you want to terminate the command.



User Tips

User Tips are used in the modules to help you complete models 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. Do not 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 this User Tip now, it is here for an example only.)

USER TIP: When an Inventor pull-down has a small triangle at the end, there is a flyout menu associated with it. A check mark means that the item is enabled. When there is no check mark, it is disabled.

WORK ALONG

Workalongs are there to help you to understand how to use a command or a series of commands to complete models and drawings using Autodesk Inventor. Complete all the steps in each workalong to complete the model or assembly to practice using the command(s) taught in that module. Try to understand how each command works and use those principles to complete the lab exercise on your own.

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

WORK ALONG Drawing Sketches with Circles

Step 1

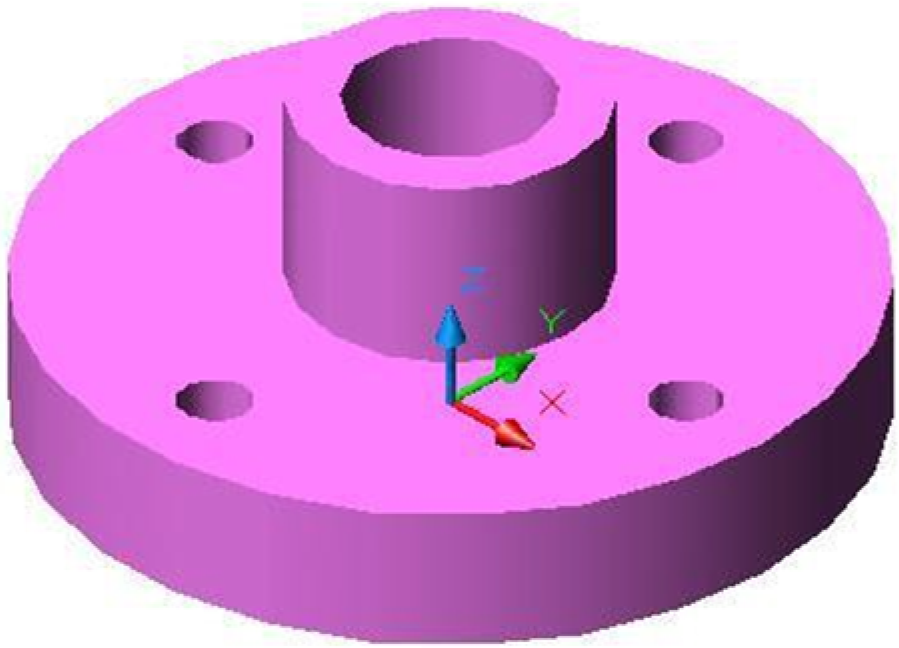
Check the default project and if necessary, set it to Inventor Course.

Step 2

Enter the NEW command to start a new part file using the template: English-Modules Part (in).ipt.

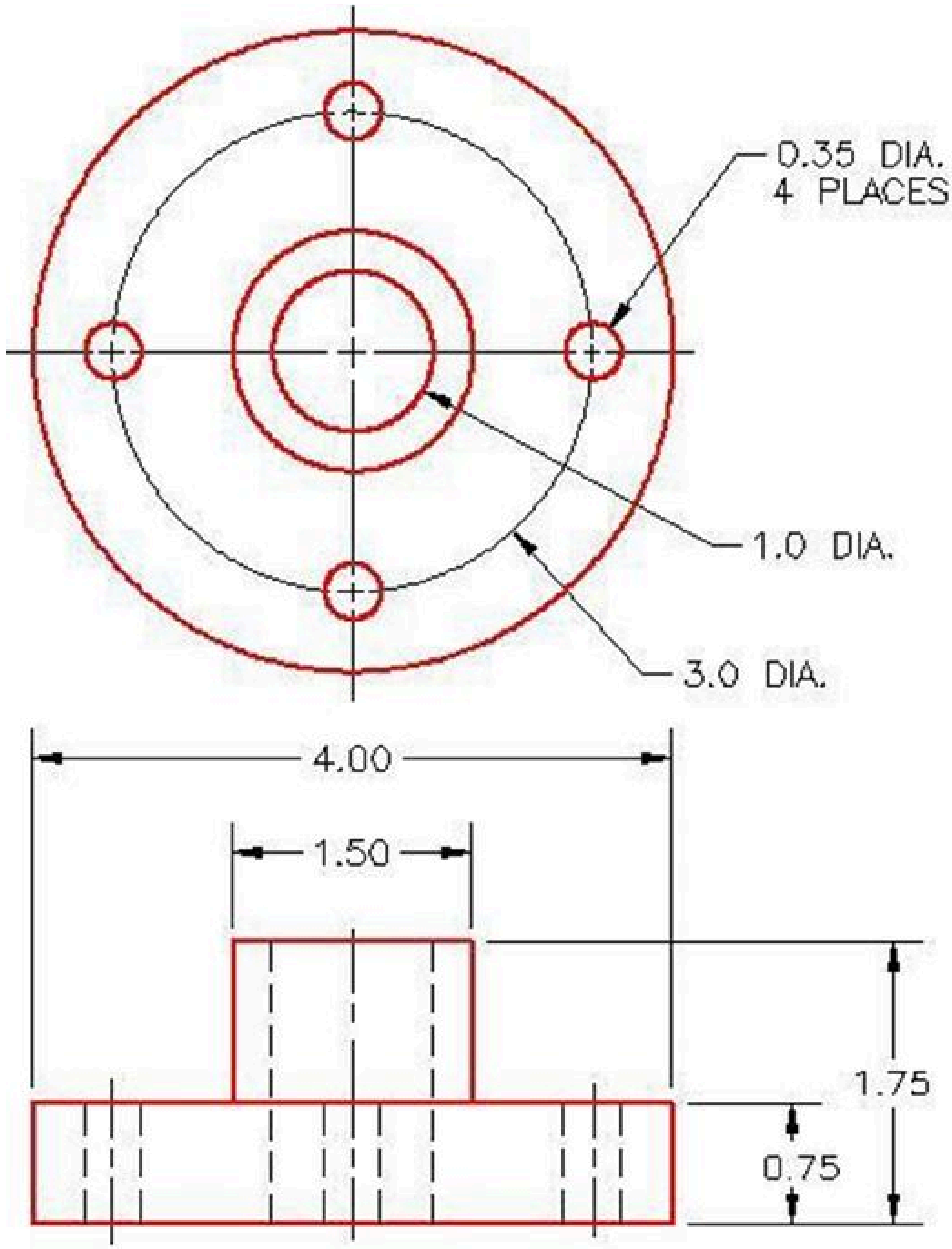
Step 3

Save the file with the name: Inventor Workalong 12-1.



Step 4

Draw the base sketch on the top view. Since this is the XY plane (default plane), use Sketch1.



Dimensioned Multiview Drawing

Geometry and Drafting Lessons

Some modules contain *Drafting Lessons* and/or *Geometry Lessons*. They are there to teach you 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 item 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. You should ensure you read them before you attempt to complete the lab exercises. For example:

(Do not attempt to understand this Geometry Lesson now, it is here for an example only.)

Geometry Lesson: Circles

A *circle* is defined as a closed curve in which all points on the circle are the same distance from its centre point. The *center point* is a single XY coordinate. Study Figure 12-1 for a detailed description of the geometry of a circle.

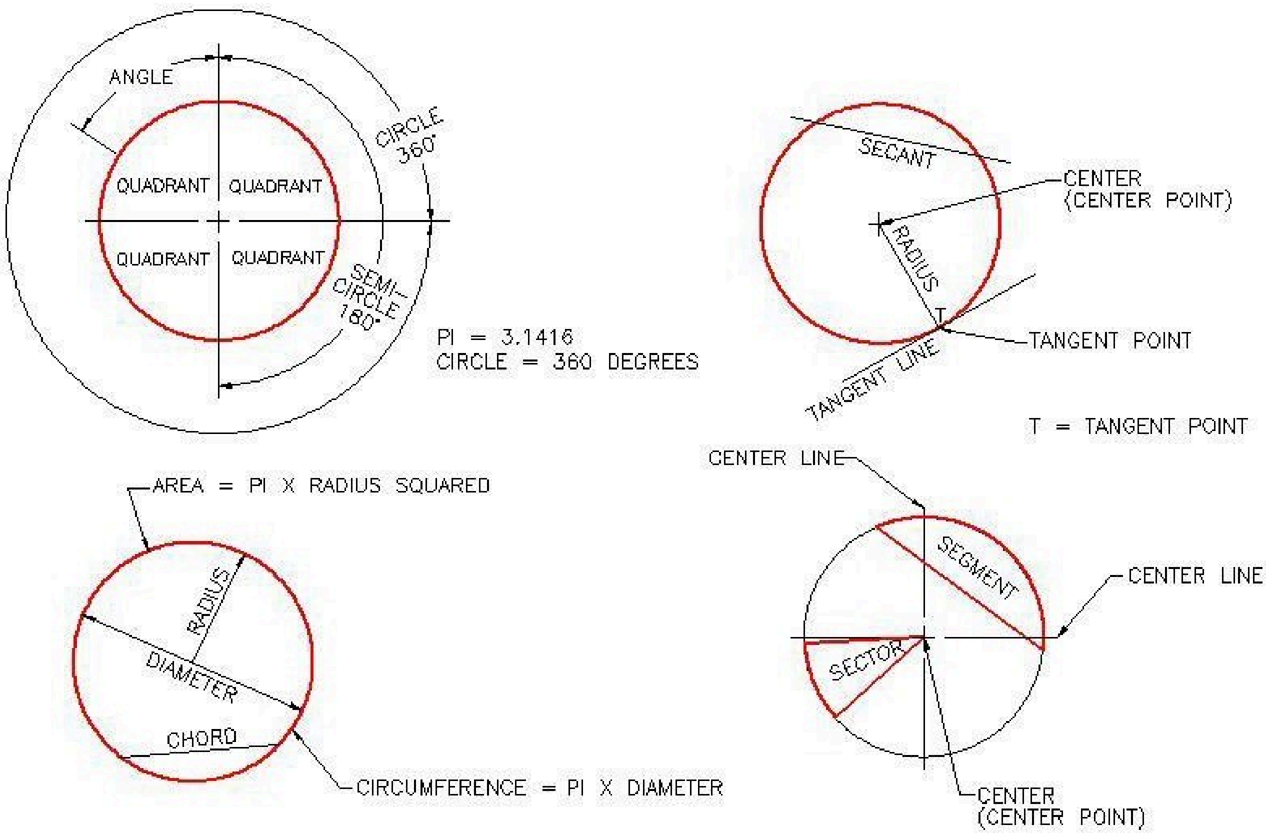


Figure 12-1
Geometry of a Circle

Key Principles

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

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

Key Principles in Module 4

1. The @ symbol means “The last absolute coordinate location”.
2. To close the last line of a series of lines, use either C (Close) or the absolute coordinate of the first point.
3. To delete existing objects on a drawing, you can either use the ERASE command or the Delete key on the keyboard.
4. Objects can either be selected before or after the command is entered.

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 model or assembly 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 have mastered the module.

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

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 four modules that you just completed. If there are any parts of this module that you have trouble completing or you cannot complete in the time allowed, you should 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 you understand the material.

Words in Red Italics

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

Construction objects are objects that are drawn in the sketch to assist the operator in completing the sketch but will be ignored by Inventor when the sketch is extruded or revolved.

Command Names

Command names are always in uppercase. For example:

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

Author's Comments

There are author comment's throughout the modules. For example:

AUTHOR'S COMMENTS: Note the location of X0Y0Z0 icon. The icon's vertex is located at X0Y0Z0. The red arrow points in the positive X direction, the green arrow points in the positive Y direction and the blue arrow points in the positive Z direction.

Key Principles in Module Introduction 1

1. To complete the lab exercises in this book, you must have the template files that accompany it, on your hard disk drive.
2. Do not skim through the modules. You must read and understand everything in each module. Do not jump around inside the module or from module to module. Work your way methodically through each module, page by page.
3. The Inventor Self-paced book's modules were written as competency-based bite-size pieces to allow you to work at your own pace and learn to use Inventor. Do not go onto the next module until you understand the module you are working on and have completed the lab exercise(s).
4. 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 model or assembly 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 have mastered the module.

Do not memorize Inventor – UNDERSTAND it.

Have fun on your journey into the fascinating, never-ending world of learning and mastering Inventor.

Configuring Your Inventor Software

Learning Outcomes

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

1. Configure your Inventor software for the Inventor book.

WORK ALONG: Configuring your Inventor Software for the Inventor book

Complete all of the following steps:

Step 1

Start Inventor. Your screen should appear similar to the figure. Your colors may not be the same, but the basic configuration should be. If any additional dialogue boxes or windows open, close them. (Figure Step 1)

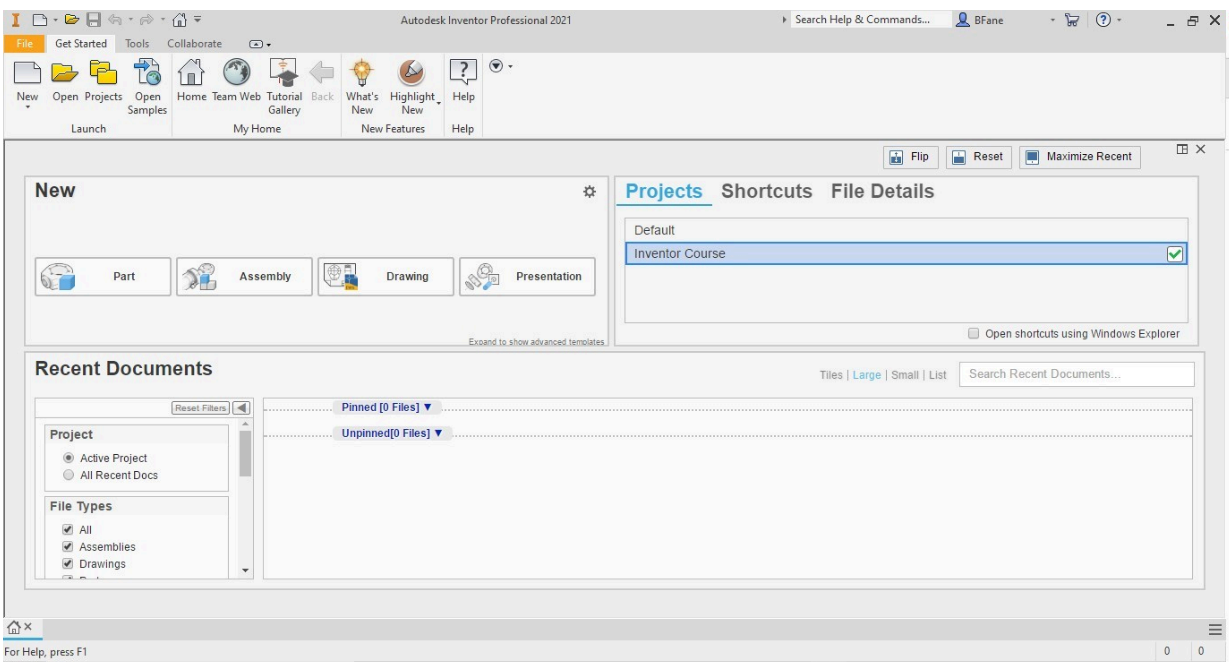


Figure Step 1 [Click to see image full size]

AUTHOR'S COMMENTS: Inventor 2020 and 2021: Your screen might differ slightly from the one shown in the figure; colors in particular. Inventor 2019 and earlier: The basic header area will be quite similar, but the main screen area will be empty. Similarly, there may be slight differences in the colors and appearance of interface elements throughout this book.

Step 2

Click the arrow icon at the end of the Ribbon menu (beside the Help ? icon). Set the pull- down menu by enabling or disabling features to match the figure. (Figure Step 2)Step 2 Click the arrow icon at the end of the Ribbon menu (beside the Help ? icon). Set the pull- down menu by enabling or disabling features to match the figure. (Figure Step 2)

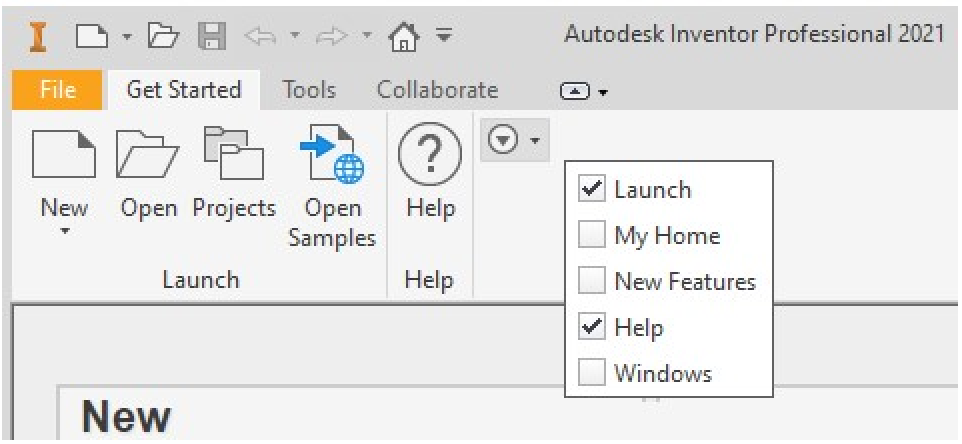


Figure Step 2

Step 3

Click the small down-arrow icon on the right side of the Pull-down menu. Enable Cycle through All. (Figure Step 3) The arrow with the box around it will now cycle you through three variants of interface display configurations.

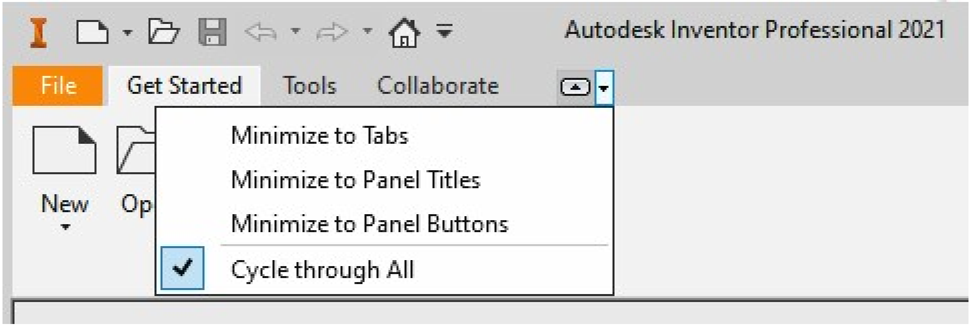


Figure Step 3

Step 4

Click the arrow icon at the end of the quick-access toolbar (by the little house icon). Make sure the enabled and disabled features match the figure. (Figure Step 4)

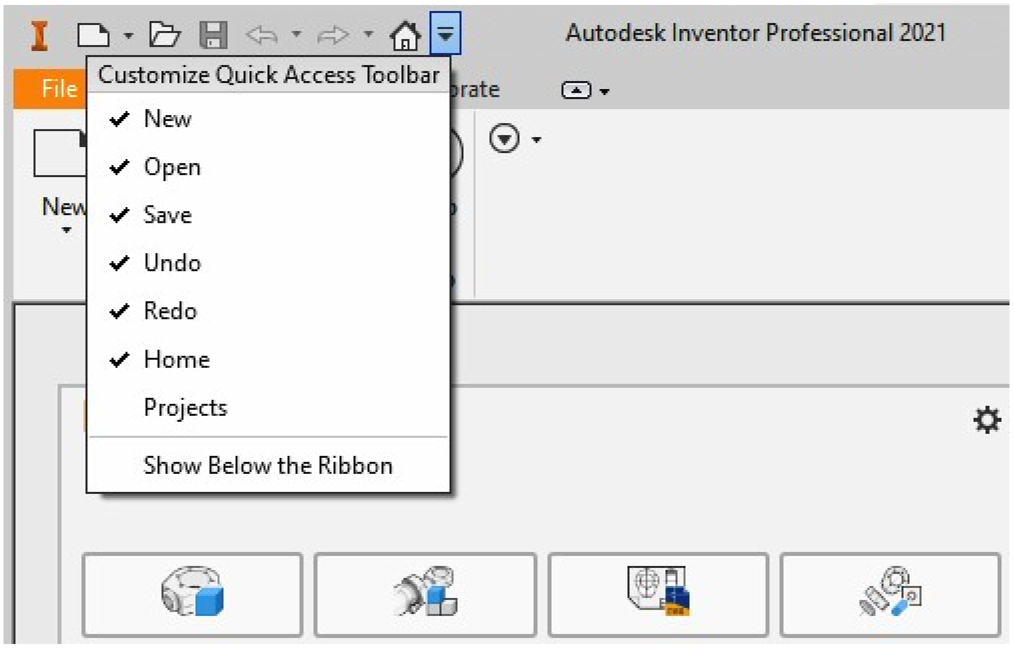


Figure Step 4

Step 5

Click *Tools* on the menu at the top of the window. Click the Application Options icon as shown in Figure Step 5. This will open the Application Options dialogue box (Figure Step 6).

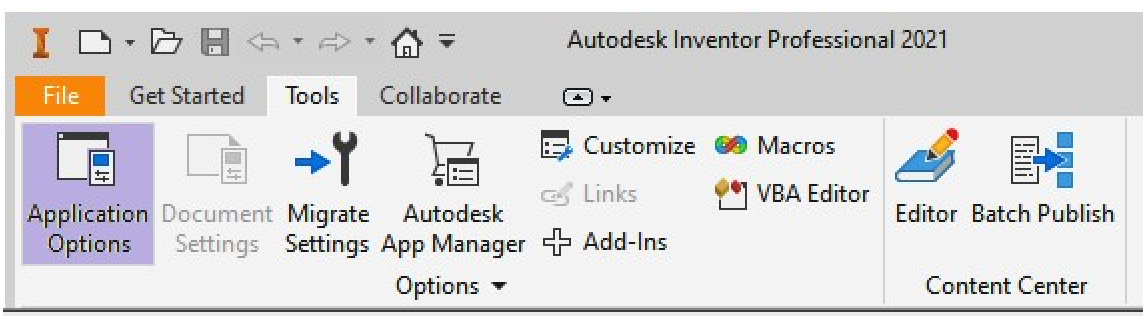


Figure Step 5

Step 6

Click on the General tab. Compare the settings of your inventor with the figure. If the settings do not match, change yours to match. (Figure Step 6)

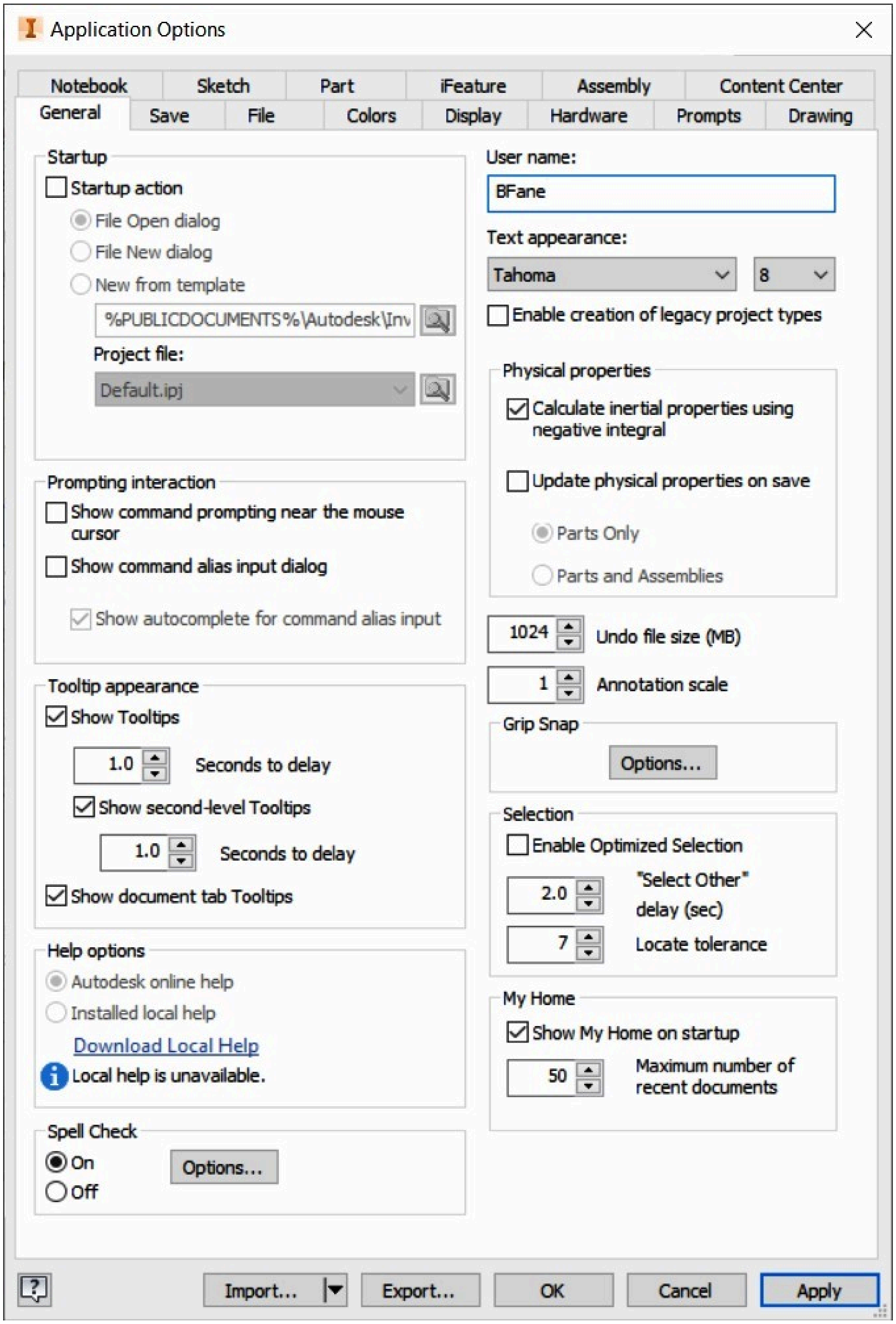


Figure Step 6 [Click to see image full size]

Step 7

Enable the Sketch tab. Compare the settings of your inventor with the figure. If the settings do not match, change yours to match. (Figure Step 7)

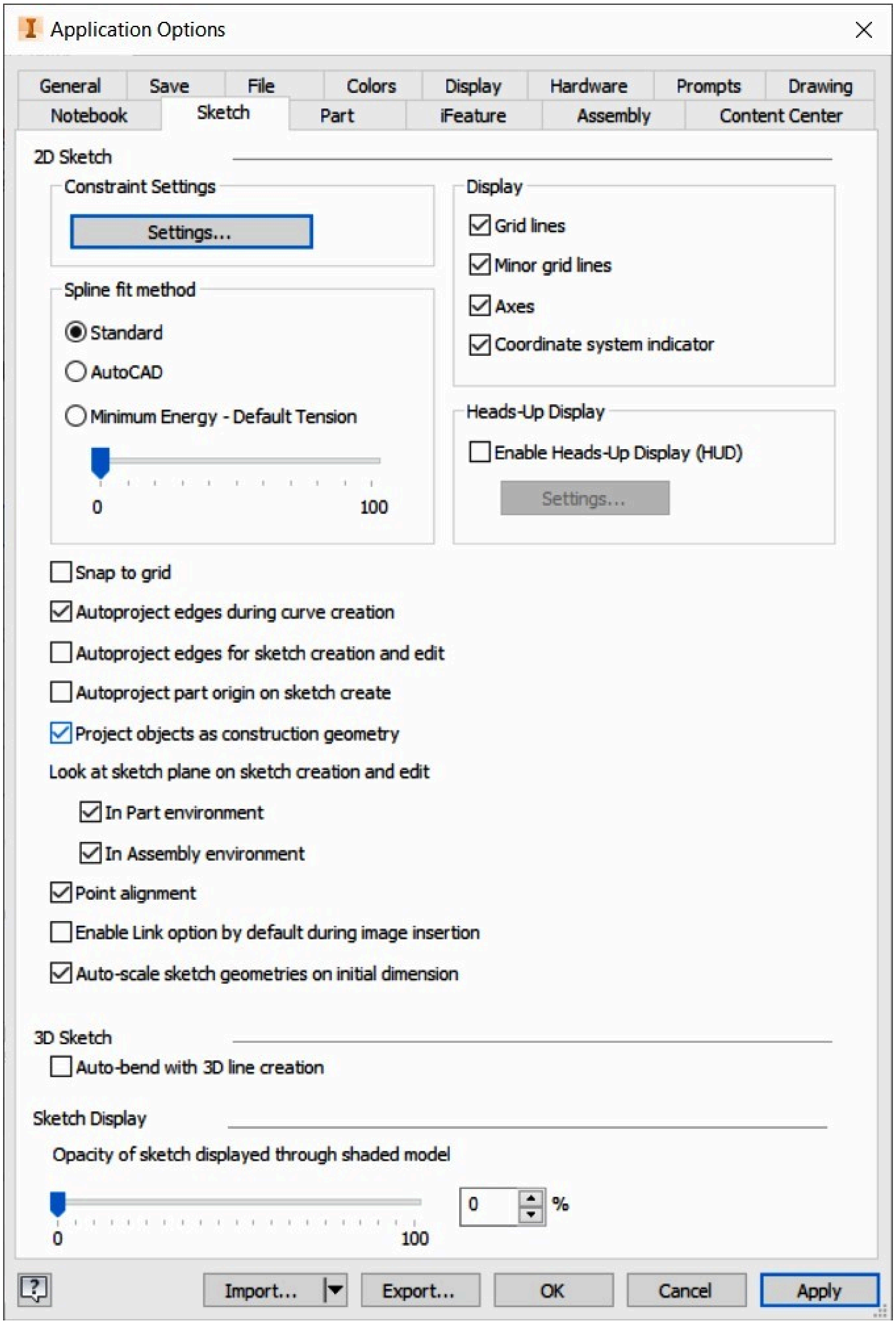


Figure Step 7 [Click to see image full size]

Step 8

Enable the Part tab. Compare the settings of your inventor with the figure. If the settings do not match, change yours to match. (Figure Step 8)

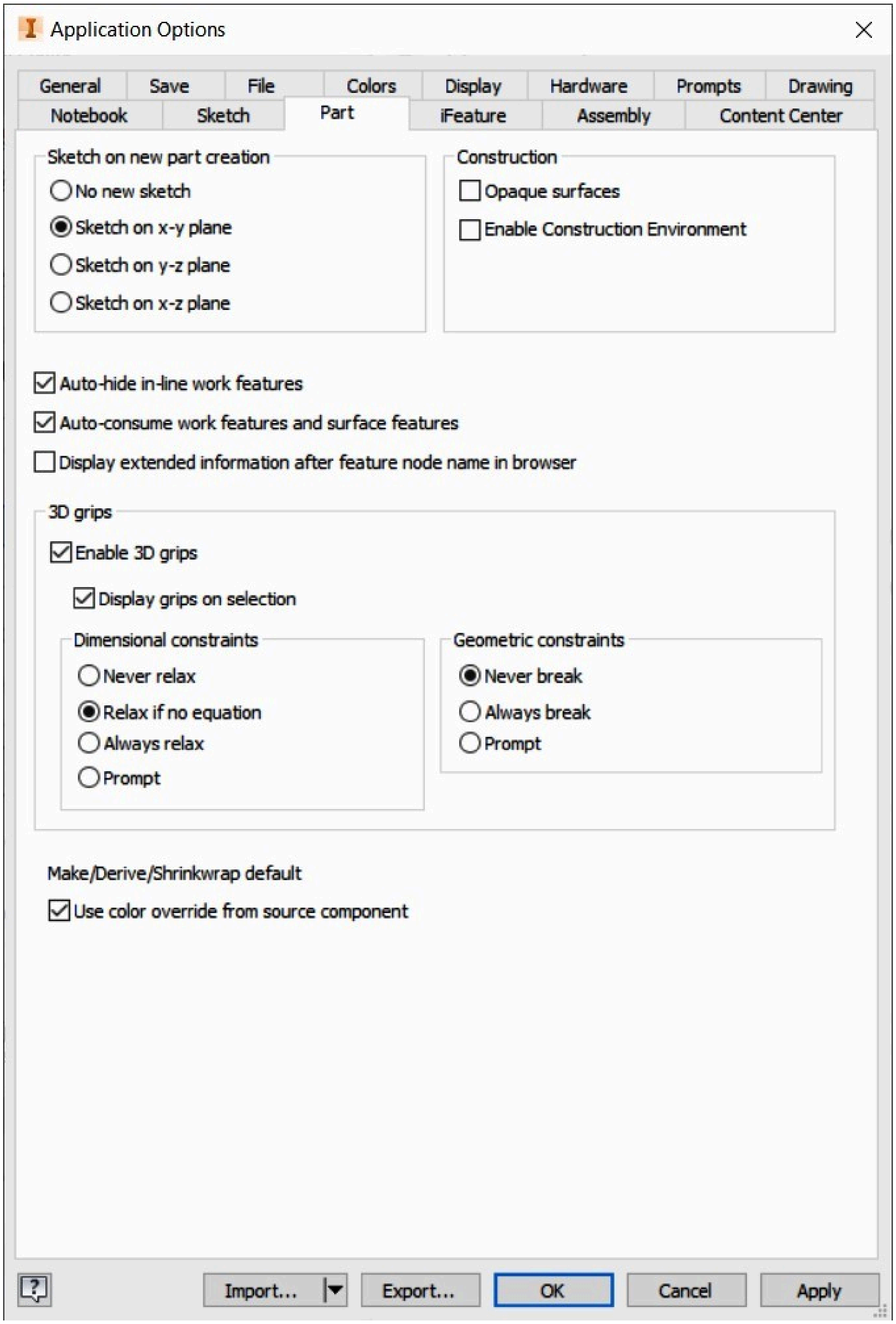


Figure Step 8 [Click to see image full size]

Step 9

Enable the Display tab. Compare the settings of your inventor with the figure. If the settings do not match, change yours to match. (Figure Step 9)

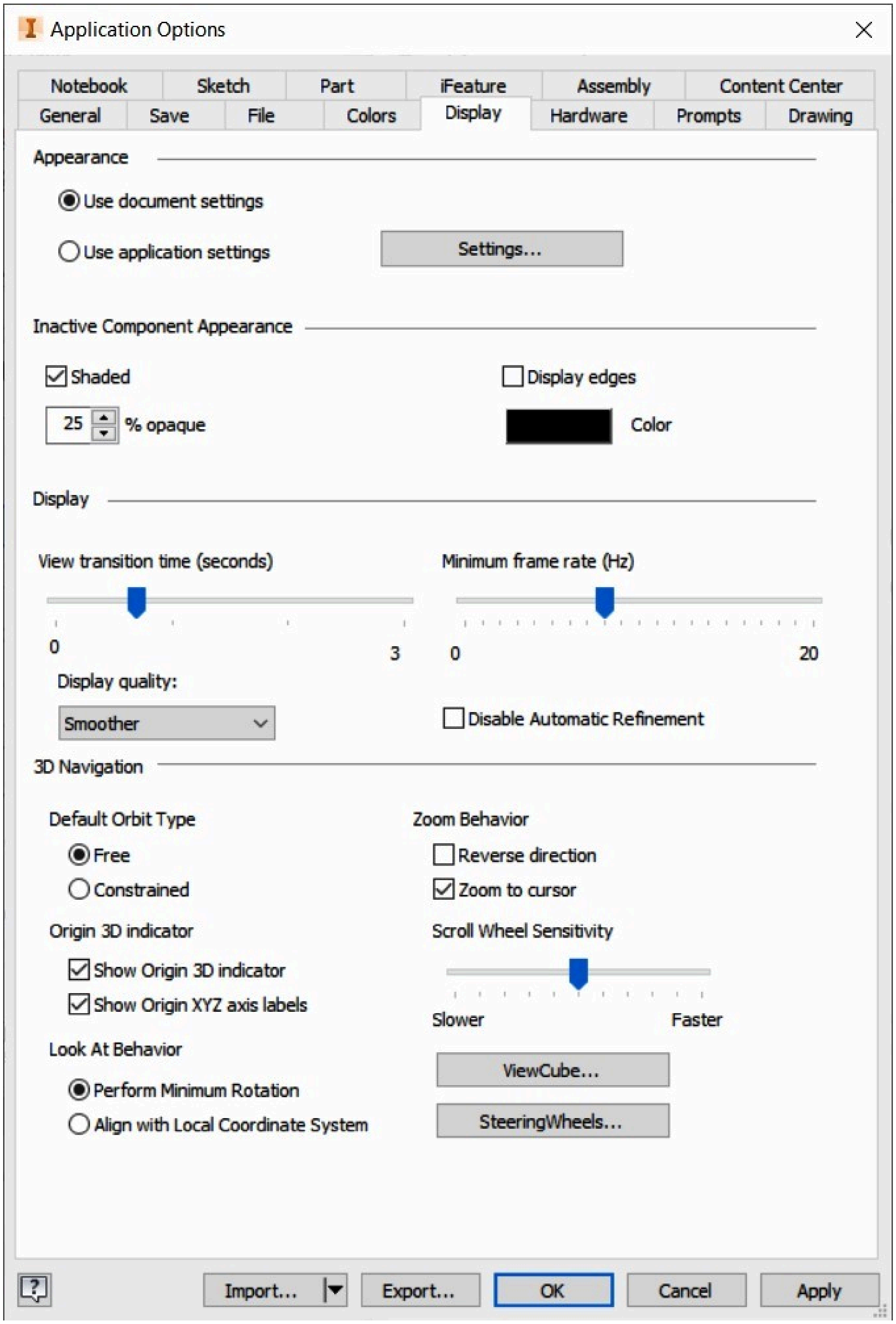


Figure Step 9 [Click to see image full size]

Step 10

In the Display tab, click the ViewCube box to open the ViewCube Options dialogue box. Compare the settings of your inventor with the figure. If the settings do not match, change yours to match. (Figure Step 10)

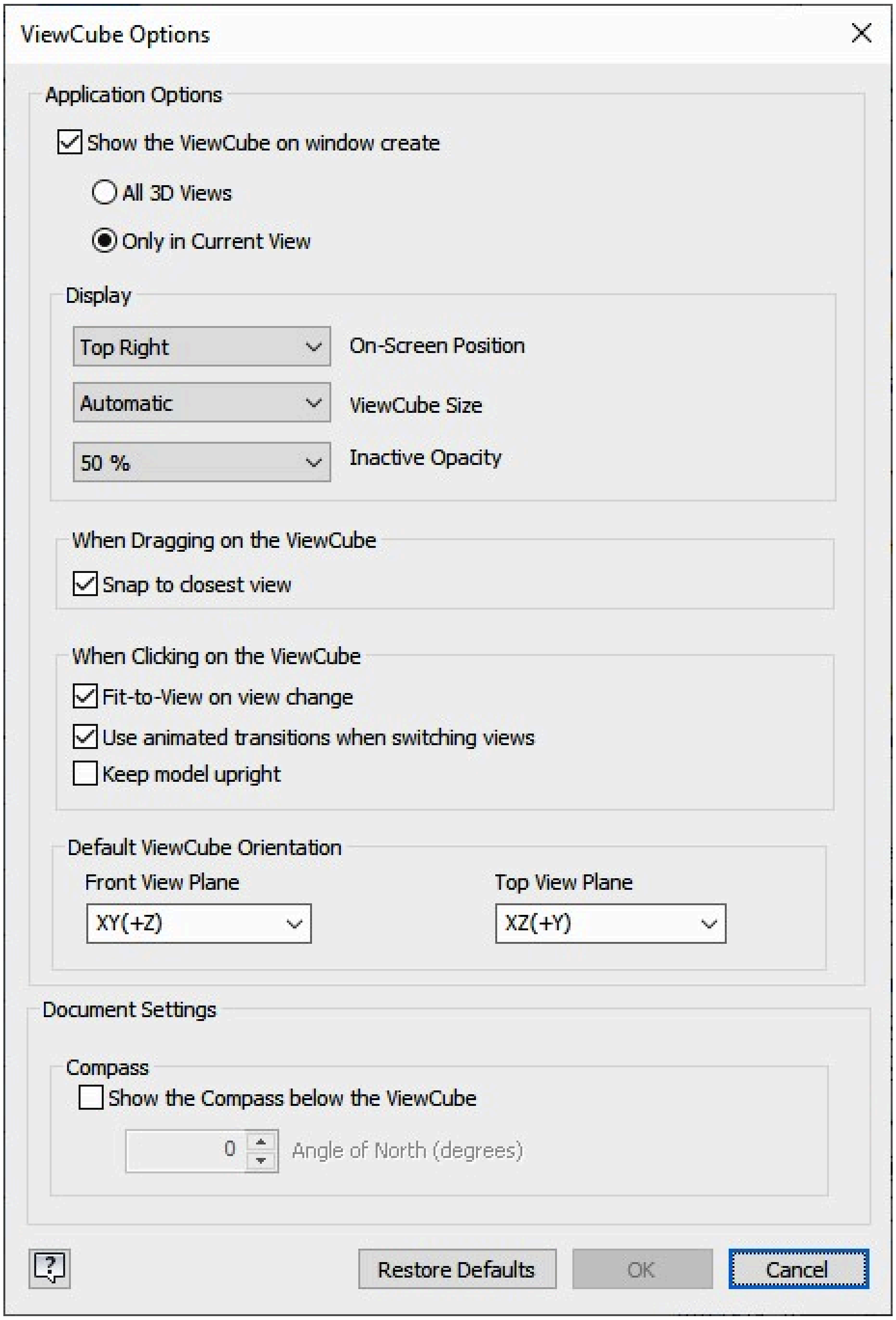


Figure Step 10

Step 11

Enable the Colors tab. Check that the settings of your Inventor match the figure.

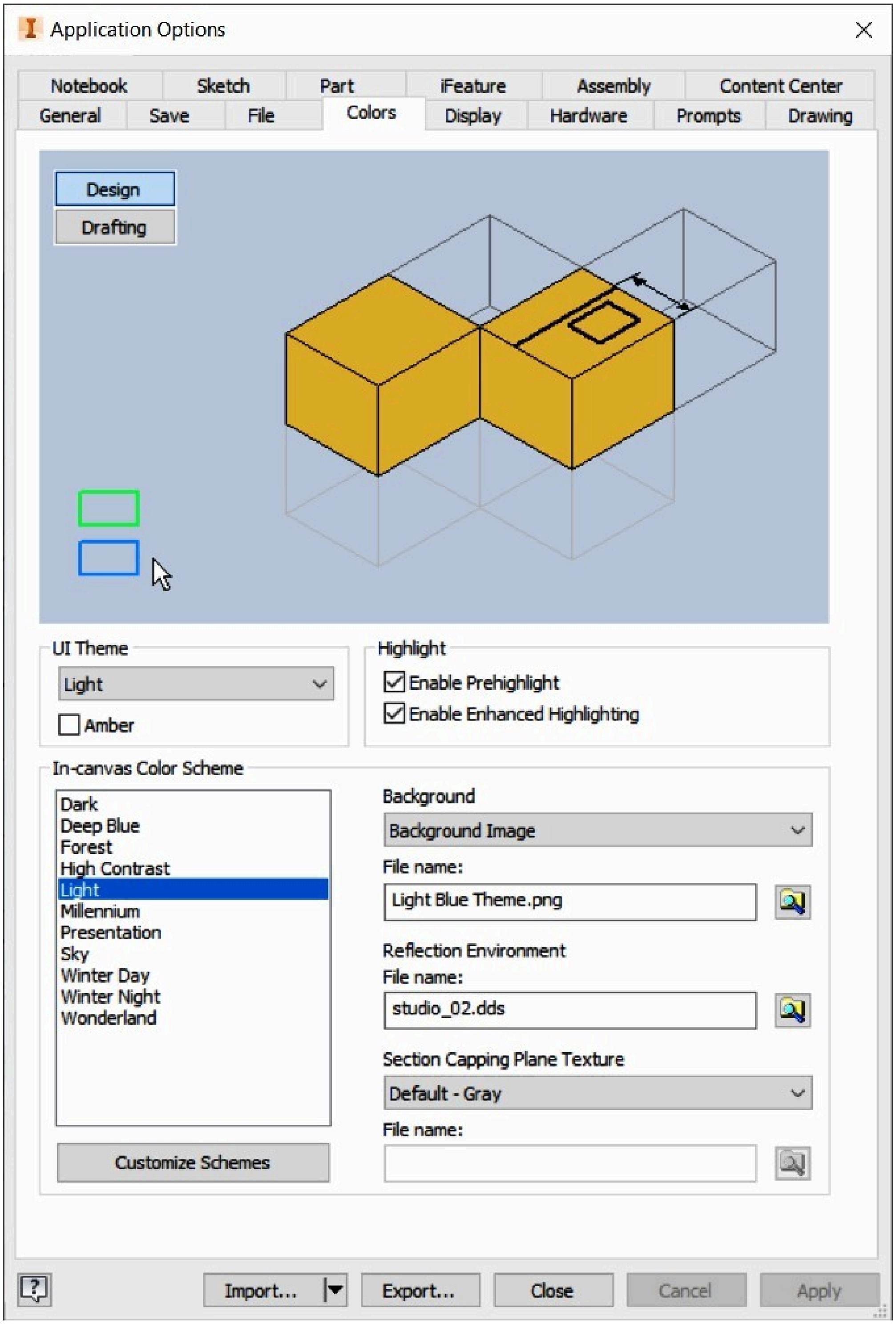


Figure Step 11 [Click to see image full size]

Step 12

Click Close button to close the Application Options dialogue box.

Step 13

Click Tools and then the Customize icon. Click the Marking Menu tab and compare the settings to your copy of Inventor.

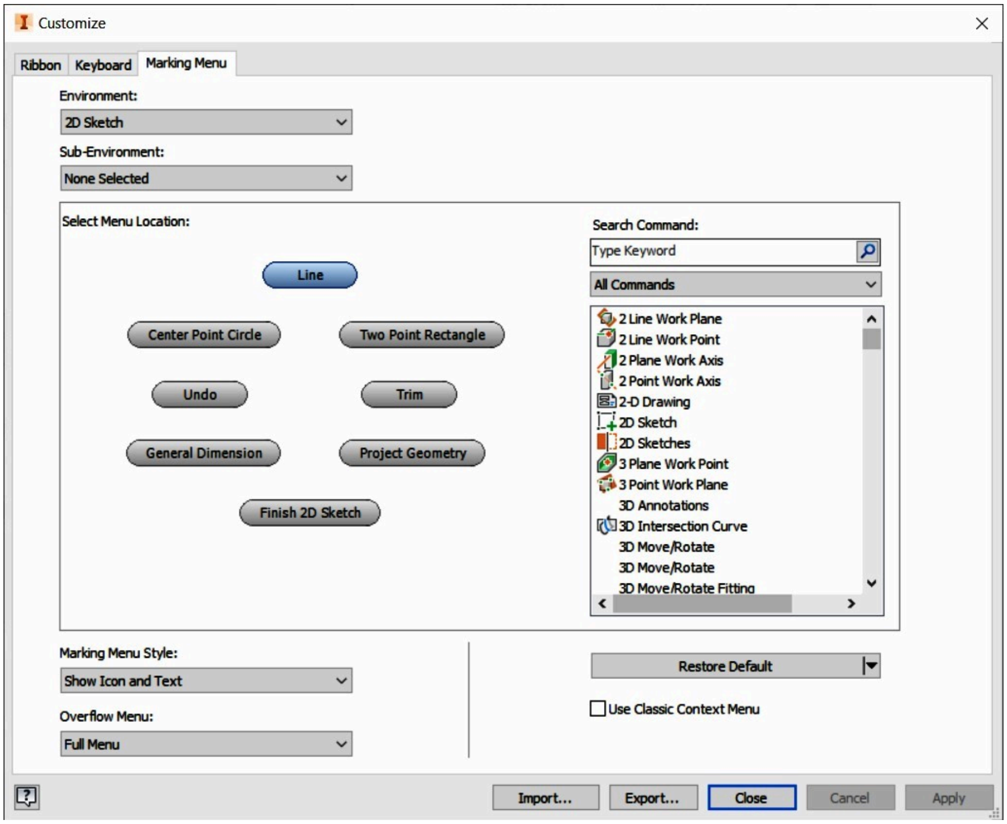


Figure Step 13 [Click to see image full size]

Step 14

Click the Inventor icon located in the top left corner and then Close to exit Inventor, **OR** click the File tab and then exit Inventor by clicking the Exit Autodesk Inventor Professional button in the bottom right corner. (Figure Step 14)

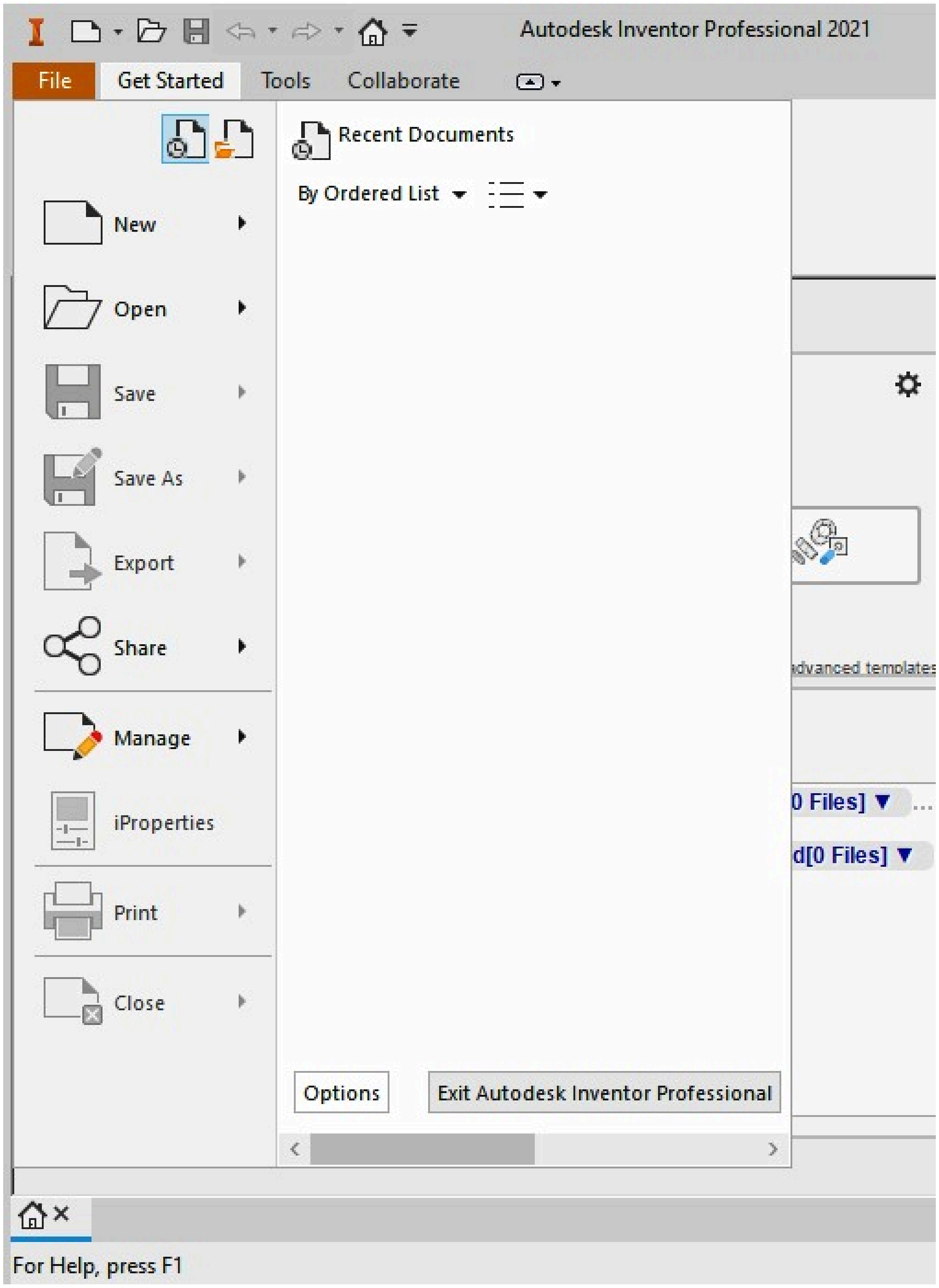


Figure Step 14

AUTHOR'S COMMENTS: The settings that you made in this module are used in the Autodesk Inventor book and are the defaults for the most part. Having completed this module you should have some idea as to how much Inventor can be customized. After you complete the book, you can configure your Inventor software to suit your personal needs.

Step 15

Go to Module 1.

Part 1

Module 1 Projects

Learning Outcomes

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

1. Describe an Inventor project.
2. Create the Inventor project that you will be using to organize and manage the files that you create while completing the Inventor book.

An Inventor Project

An Inventor *project* is a user named process to logically organize, store, and manage the valid links to the files created for an undertaking. For each project a user created name and a home folder is assigned. Inventor creates a project file that contains the project's parameters and the paths to the locations of the files in that project. There is no limit to the number of projects that can be created. Inventor assigns a shortcut for each project so that you can easily select the appropriate project when required.

A project should be created so that it has a logical connection between the files in it. For example, if you were designing and drawing an office chair, all the individual parts of that chair, the assembly drawings, the design data, and the 2D working drawings would be stored in a project that you might name ' Office Chair '.

When a project is created, Inventor automatically creates a project file and saves it in the home folder of the project. A *project file* is automatically given the extension .ipj and contains the project parameters data as shown in Figure 1-1. This file also specifies the paths to the templates and files in the project. The information, parameters and data contained in the project file can be edited as required.

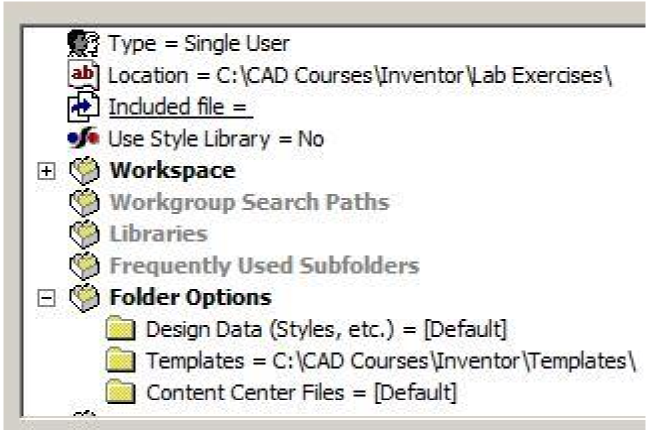


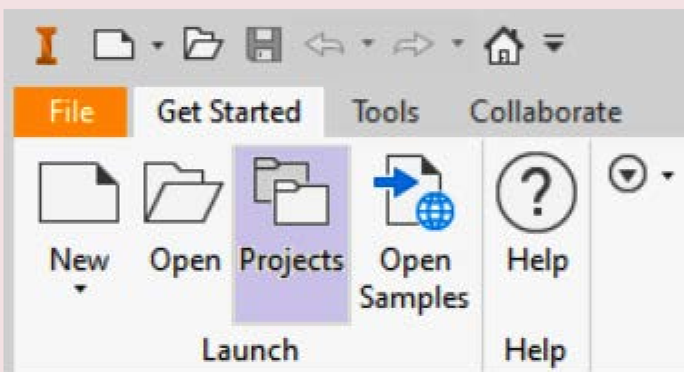
Figure 1-1
Typical Project Parameter Data List

MUST KNOW: When a project is created, Inventor automatically creates a project file and saves it in the home folder of the project. A project file is automatically given the extension.ipj and contains the data list of the project's parameters. The extension .ipj is an acronym for Inventor Project.

Inventor Command: PROJECTS

The PROJECTS command is used to create or manage Inventor projects.

Shortcut: **None**



WORK ALONG: Creating the Project for the Inventor book

Step 1

Start Inventor and enable the Get Started tab. In the ribbon menu, click Projects. This will display the Projects window. (Figure Step 1)

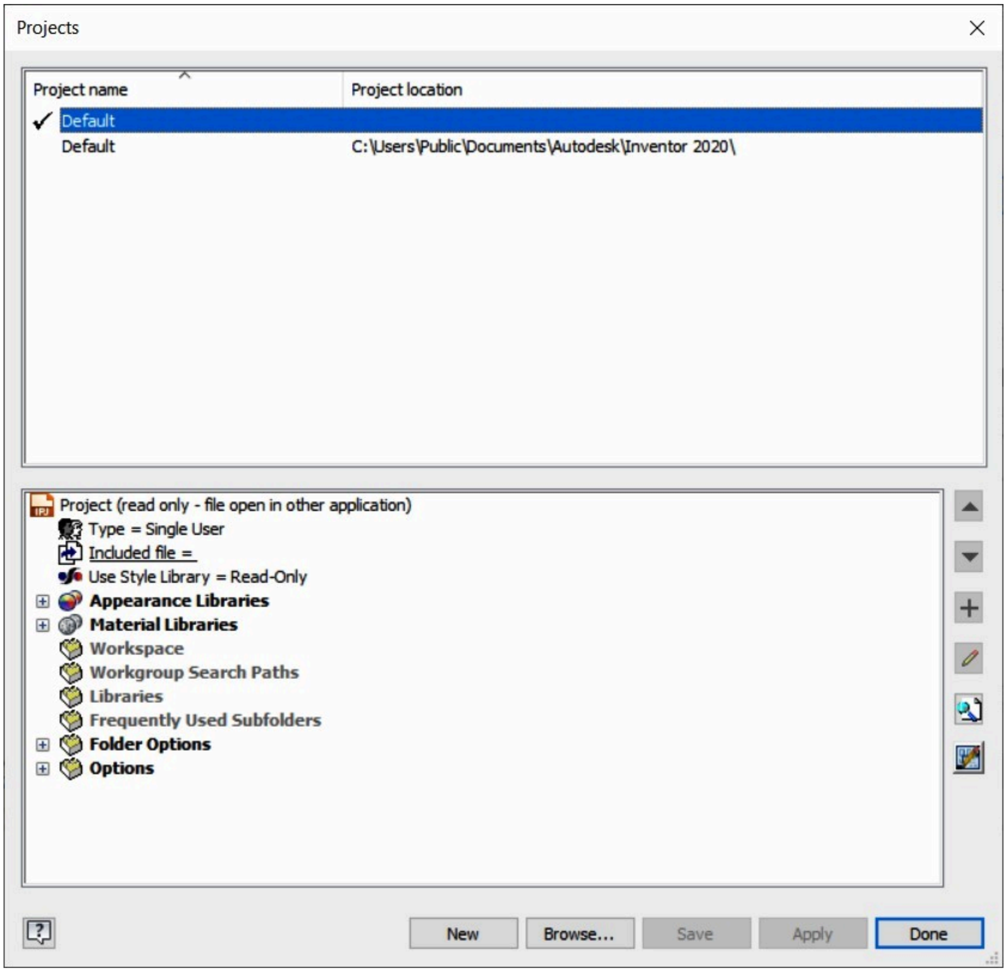


Figure Step 1 [Click to see image full size]

AUTHOR'S COMMENTS: Do not be concerned if your version of Inventor does not match the figure exactly.

Step 2

Click the New button along the bottom of the dialogue box. The Inventor project wizard window will open. Enable the New Single User Project button and then click Next. (Figure Step 2A and 2B)

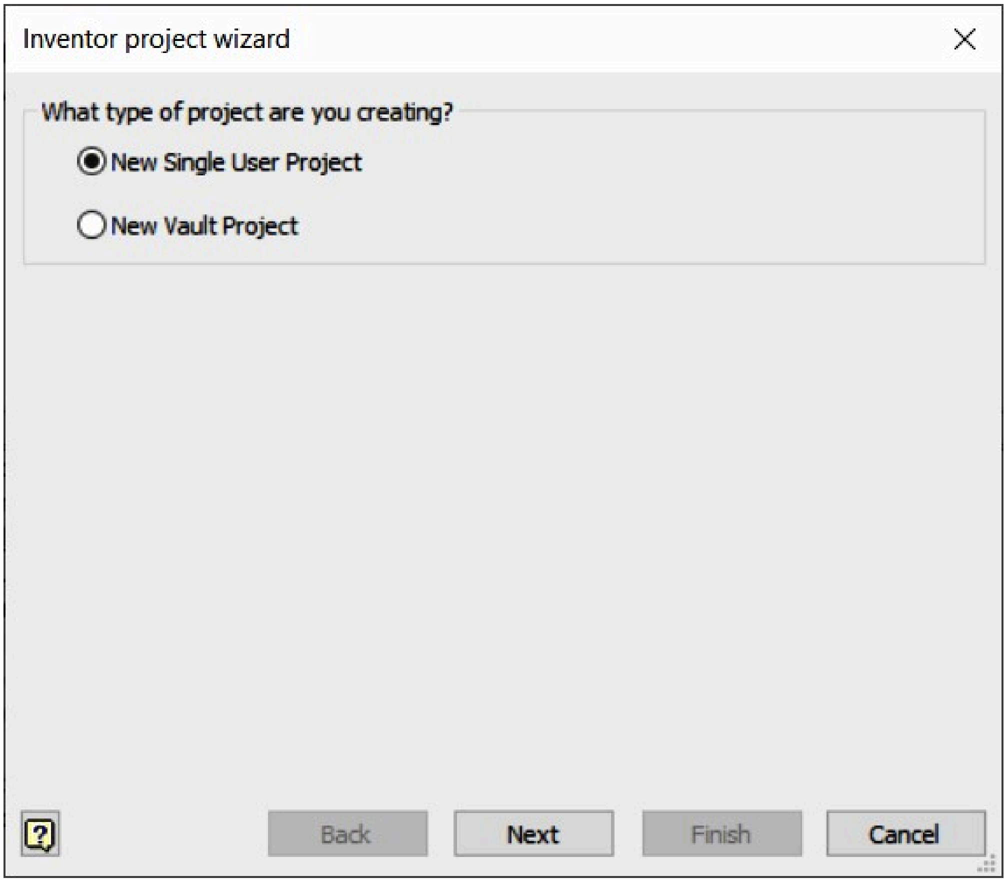


Figure Step 2A

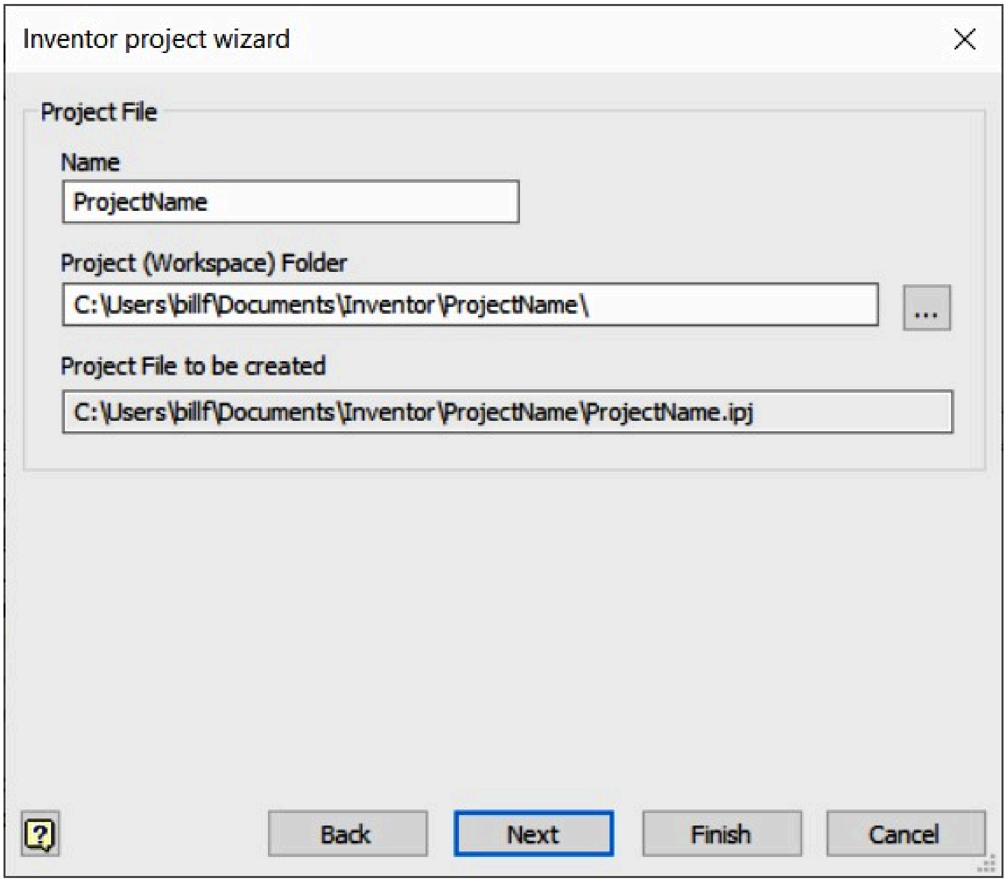


Figure Step 2B

AUTHOR'S COMMENTS: Do not be concerned if your version of Inventor does not match the figures exactly.

Step 3

Name the project: Inventor Course. (Figure Step 3)

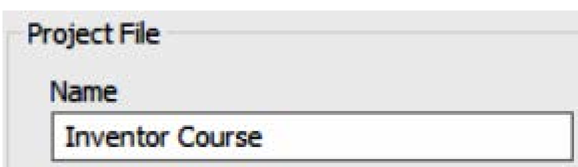


Figure Step 3

Step 4

Set the location for the Project (Workspace) Folder. To do that, click the Browse button (the one with the 3 dots at the right-hand of the Project (Workspace) Folder) window. In the Browse for Folder dialogue box, locate the folder: C:\CAD Courses\Inventor\Lab Exercises. Highlight it by selecting it and click OK. (Figure Step 4A and 4B)

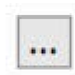


Figure Step 4A

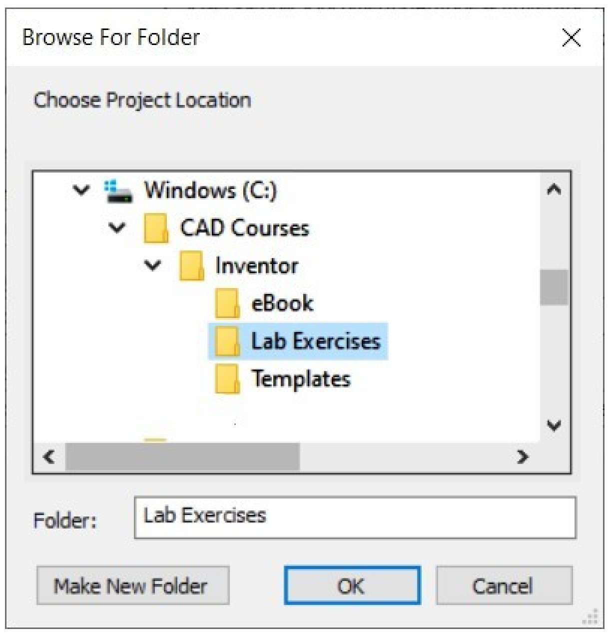


Figure Step 4B

AUTHOR'S COMMENTS: These are the folders that you created when you download the book and templates for the Inventor course. When the Inventor Course project is set as the current project,Inventor will automatically save the files that you create into the folder Lab Exercises – Inventor Course project.

Step 5

You should now see the Project (Workspace) Folder location as follows: **C:\CAD Courses\Inventor\Lab Exercises** (Figure Step 5)

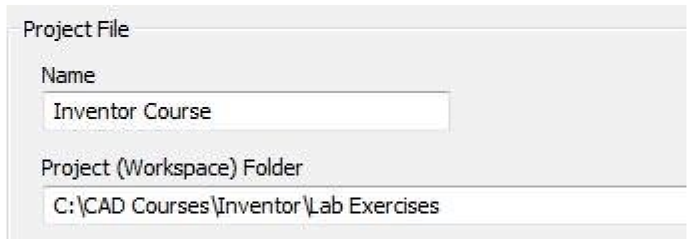


Figure Step 5

AUTHOR'S COMMENTS: C:\CAD Courses\Inventor\Lab Exercises is the Home Folder for the Inventor Course project.

Step 6

The Project File to be created will automatically be created by Inventor. (Figure Step 6)

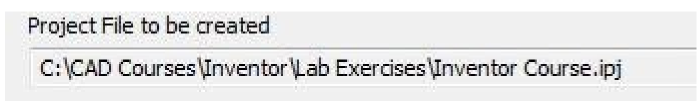


Figure Step 6

AUTHOR'S COMMENTS: Note how the box is grayed out. That means it cannot be edited and is there for information only.

Step 7

Check the completed page and ensure it matches the figure. Click the Finish button to complete the project setup. (Figure Step 7)

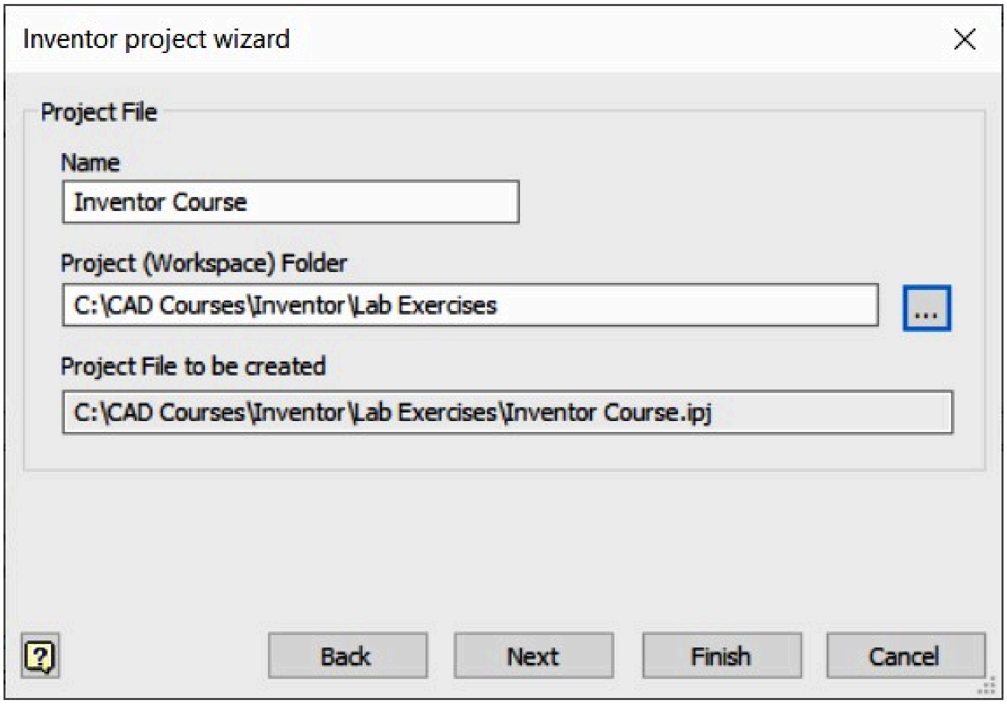


Figure Step 7

Step 8

The Project window will show the new project that you just created. Double click Inventor Course in the Projects name column and note how the check mark icon beside the Inventor Course project indicates it is the current project. (Figure Step 8)

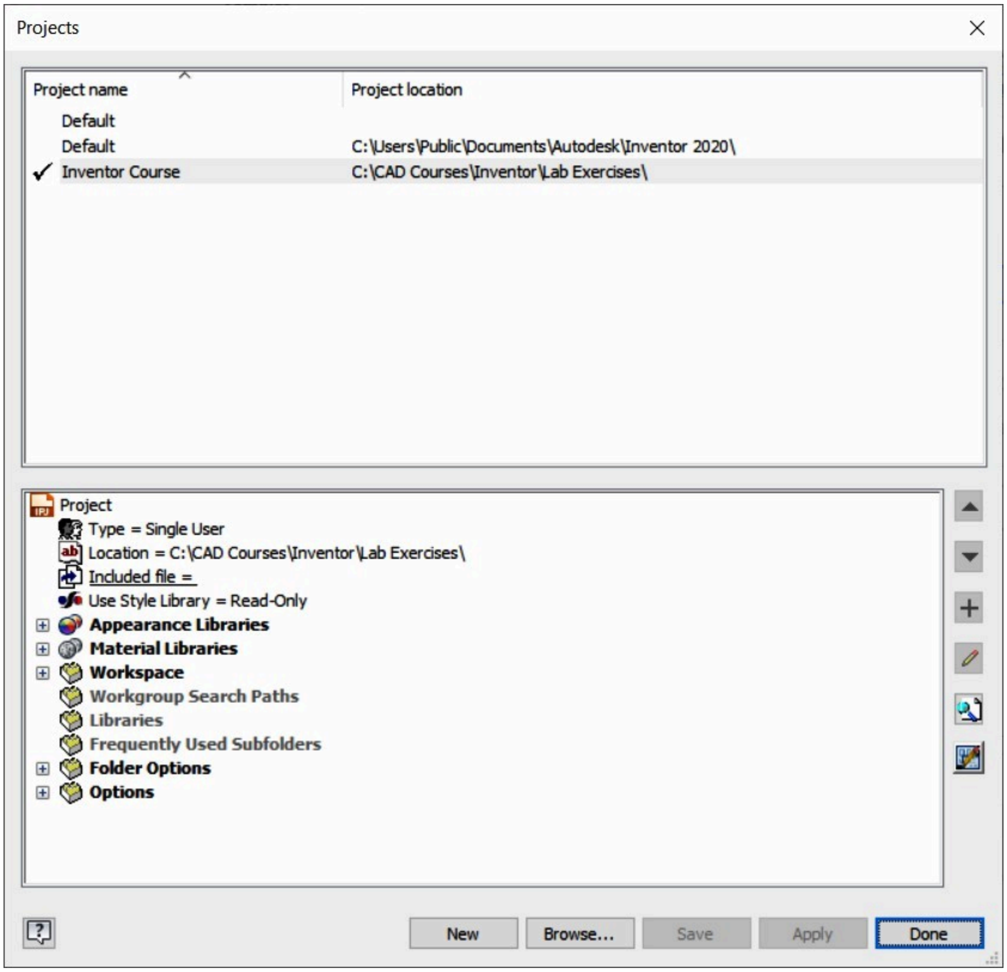


Figure Step 8 [Click to see image full size]

AUTHOR'S COMMENTS: Ensure that the *Inventor Course* project is always the current project when you are working on all exercises in the Inventor book.

Step 9

Down near the bottom, expand Folder Options and click the Templates folder to select it. (Figure Step 9)

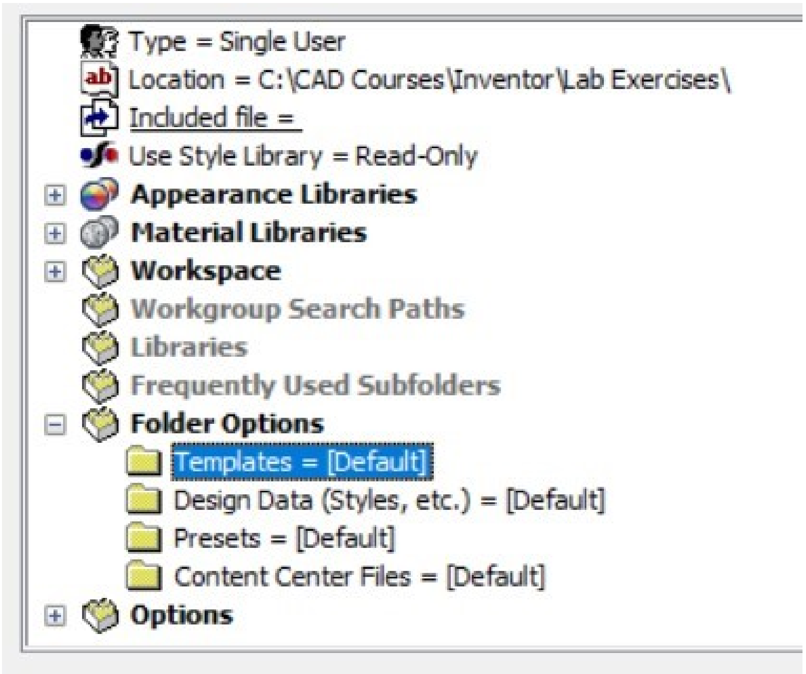


Figure Step 9

Step 10

While the Templates folder is selected, right-click the mouse. In the right-click menu, click Edit. (Figure Step 10)

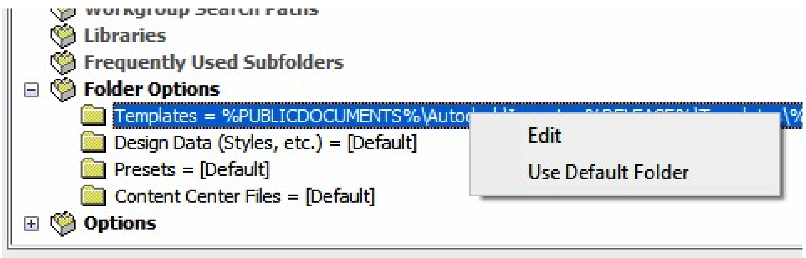


Figure Step 10

Step 11

Click the Browse icon. (Figure Step 11)

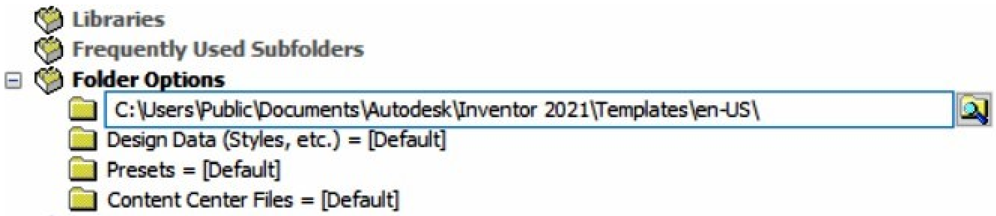


Figure Step 11

AUTHOR'S COMMENTS: The Browse icon is the small magnifying glass folder icon at the end of the box.

Step 12

In the Browse For Folder dialogue box, locate and select the Templates folder: **C:\CAD Courses\Inventor\Templates** (Figure Step 12)

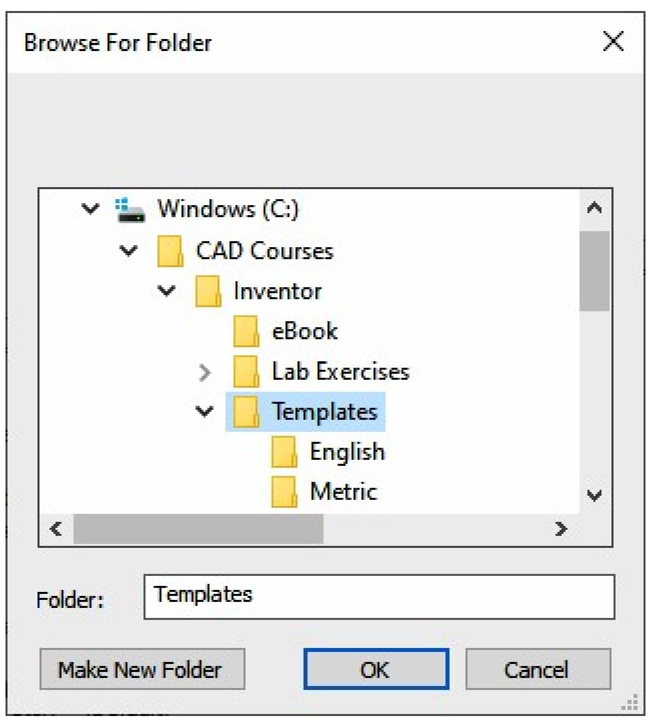


Figure Step 12

MUST KNOW: The folder location for the templates in the Inventor book should be set to C:\CAD Courses\Inventor\Templates\ in the Inventor Course project.

Step 13

Click OK and the dialogue box will appear similar to the figure. (Figure Step 13)

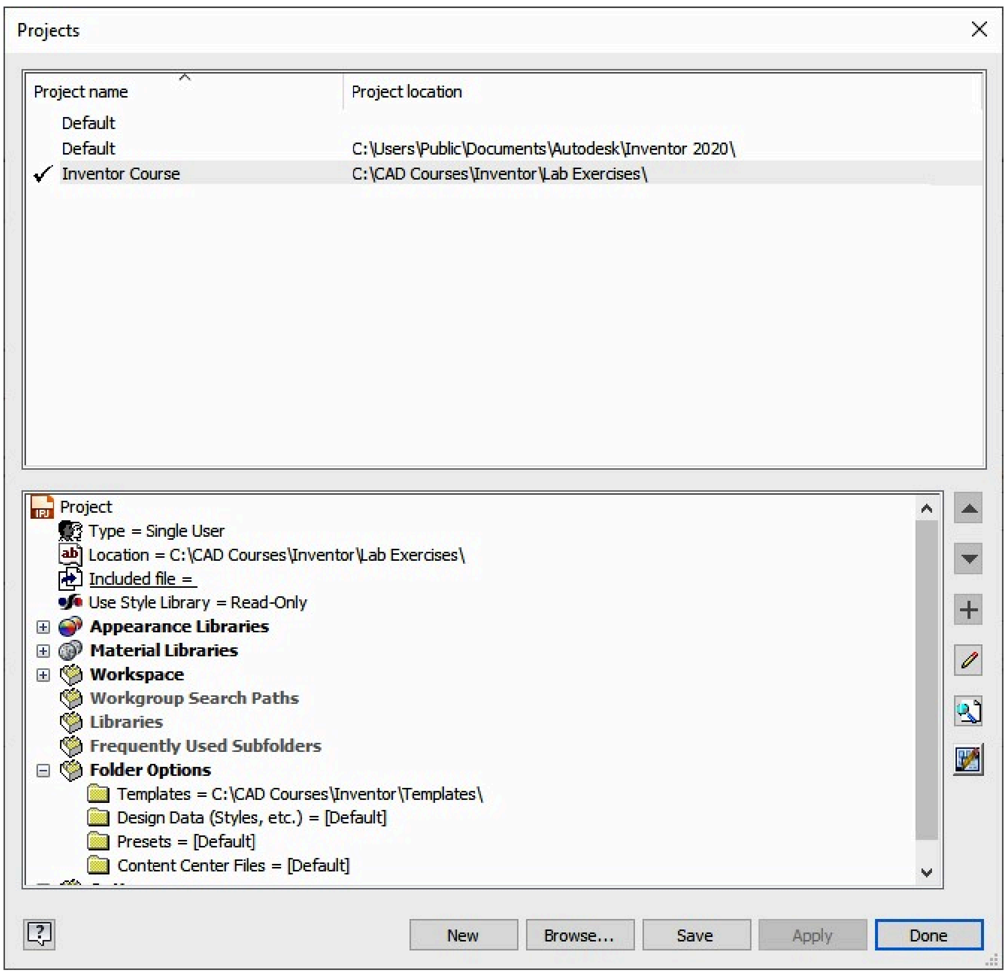


Figure Step 13 [Click to see image full size]

AUTHOR'S COMMENTS: This is the folder location where Inventor will look for template files when you enter the NEW command.

Step 14

Click Apply and Done to complete the new project.

Step 15

Exit Inventor.

Step 16

Open Windows Explorer and locate the folder Lab Exercises in the folder list. The file Inventor Course.ipj, that was automatically created by Inventor in this workalong, should appear in the file list. (Figure Step 16)

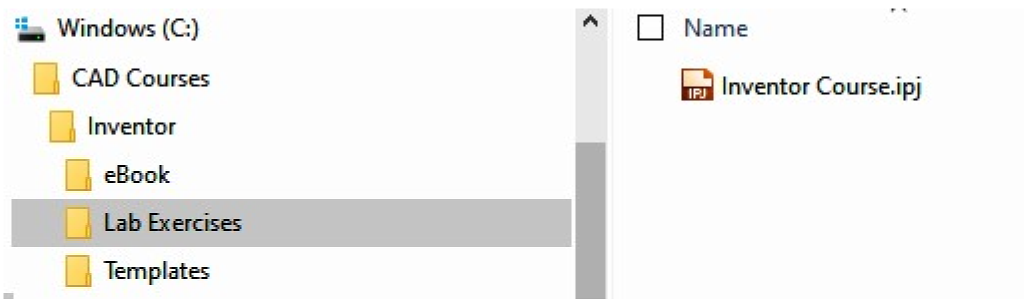


Figure Step 16

MUST KNOW: In the Projects window dialogue box, Inventor Course should always be the current project when you are working on all exercises in the Inventor book.

Key Principles

Key Principles in Module 1

1. An Inventor project is designed to logically organize, store, and manage the valid links to the files

created for each undertaking. For each project, a name and a home folder must be assigned.

2. A project file is automatically given the extension .ipj and contains the project's specifications.
3. check mark beside the Inventor Course project means it is the current project. Ensure that this project is always the current project when working on all exercises in the Inventor book.

Module 2 Inventor's User Interface

Learning Outcomes

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

1. Describe template files, part files, sketches, and 3D solid models.
2. Describe and configure Inventor's user interface, its menus, and use of the mouse.
3. Apply the NEW, SAVE, and CLOSE commands to create a new part file using a template plus name, save, and close a file.

Inventor's Graphic Window

When a file is active, Inventor will display the Graphic window. The Graphic window has two different modes, the Sketch mode and the Model mode. The *Sketch mode* is used to create and edit 2D sketches that are then extruded or revolved to create 3D solid models. The *Model mode* is used to view, manipulate and modify 3D solid models. You can switch between these two modes to construct or edit and create the solid model. The mode that is currently displayed is called the current mode. The Model mode must be the current mode to save the file.

A 3D Solid Model

A *3D solid model*, Figure 2-1, is the best possible computerized representation of a real object. A solid model can be colored or rendered plus the mass properties can be obtained from it. Mass properties are attributes such as volume, weight, surface area, moments of inertia, and centre of gravity. They are taught in Module 20.

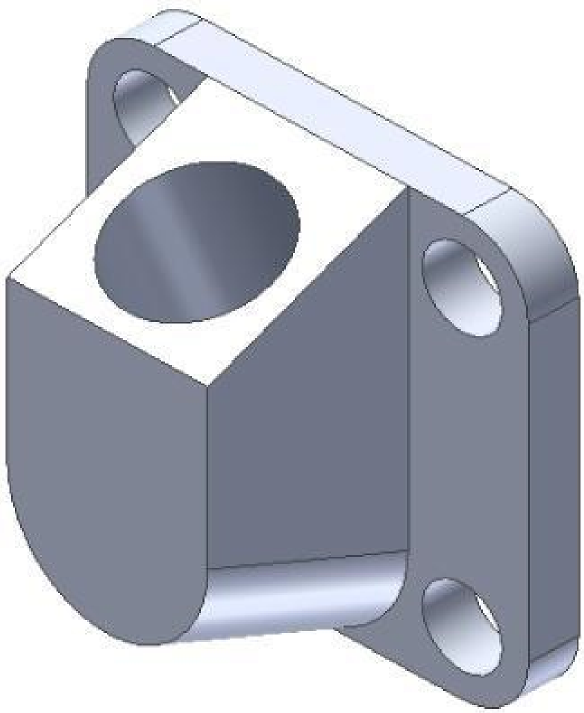


Figure 2-1
A 3D Solid Model

An Inventor File

Regardless of the type of files that are created using Inventor, they are called files as compared to files created on a CAD system which are typically called drawings. A *file* can be a 3D solid model (called a part), an assembly, a presentation, or a drawing created in Inventor and saved in digital format. The four types of files, taught in this book, that Inventor can create and save are part files, assembly files, presentation files, and drawing files.

A Part File

A *part* file is one 3D solid model. A part file has the file extension .ipt which is an acronym for Inventor ParT. A part file can be used on its own, used to create a working drawing, used as part of an assembly file, or used as part of a presentation file. Assembly and presentation files are taught in Module 21 to Module 26.

Starting a New File

A new file is started using the NEW command. The NEW command forces you to select the template file that will be used to create the new file. Every new Inventor file must be created from an existing template file. The file currently being editing in Inventor is called the *current file* or sometimes the *active file*.

Templates

A *template* file is simply an Inventor file, set with the desired parameters by its creator, named, and saved. As part of the Inventor book there are two distinct sets of templates provided. One set is in english units and the other set is in metric units. The English templates use the base unit of inches and the metric templates use the base unit of millimeters. Template files can also contain modelling objects so that several similar parts can easily be started from a common pre-built unit.

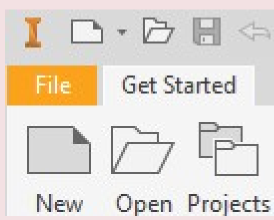
Saving Files

Inventor keeps the current file in RAM memory. If the computer cashes or the power fails while the operator is working on a file, all the work on the file, since the last time it was saved, will be lost. When the file is saved it saves the current file that is in RAM memory onto the disk drive.

Ensure that the current file is saved frequently to avoid losing production time. You should get into the habit of saving the current file every 5 to 10 minutes.

Inventor Command: NEW

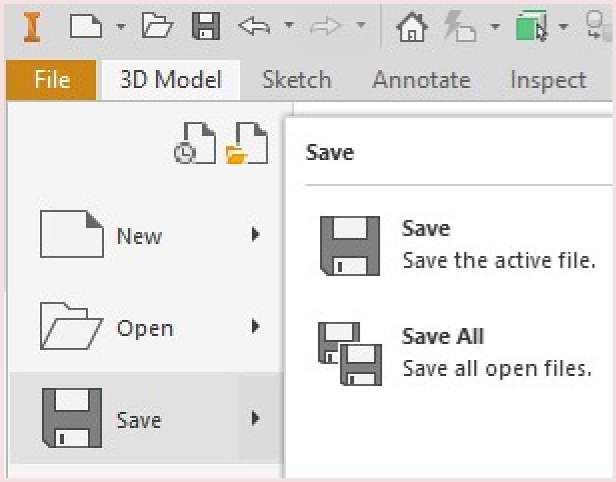
The NEW command is used to start a new file.



Inventor Command: SAVE

The SAVE command is used to save the current file from RAM memory onto the disk drive.

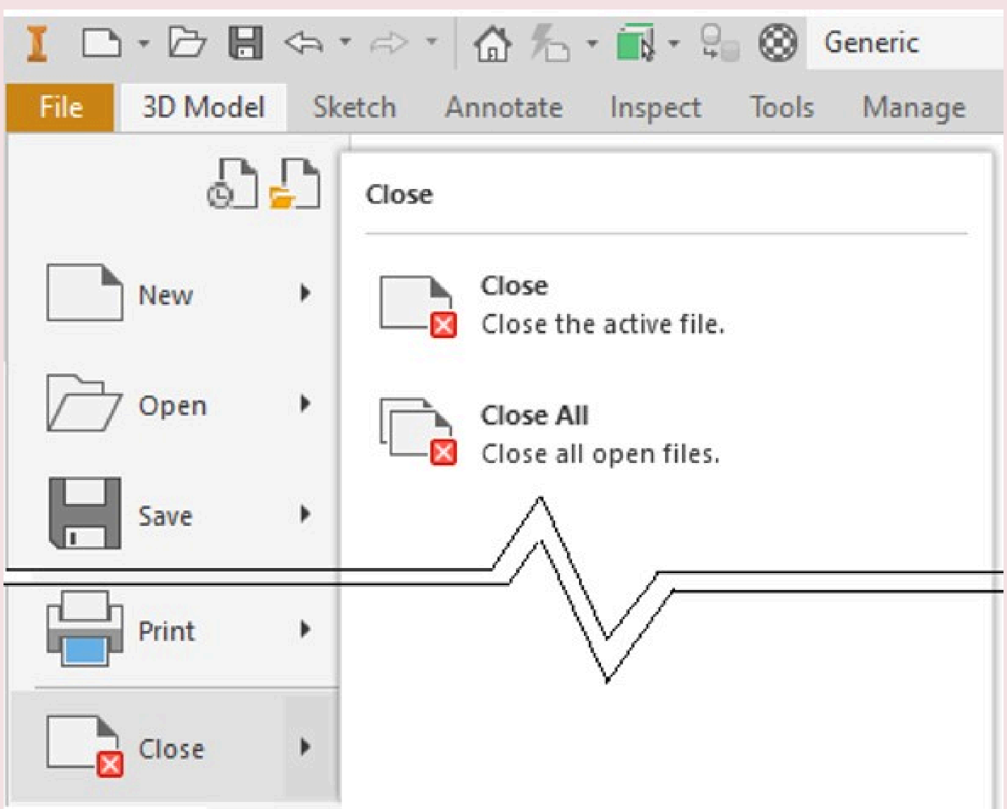
Shortcut: **CTRL-S**



Inventor Command: CLOSE

The CLOSE command is used to close the current file.

Shortcut: **None**



Inventor's Menus

Inventor has many different menus that are used by you to give instructions to Inventor while constructing and editing files. The Inventor menus taught in the Inventor book are the Pull-down menu, the Ribbon menu, the Quick Access toolbar, the Browser bar, and the Right-click menu.

Inventor's Pull-Down Menu

To pull down Inventor's Pull-down menu, click the File tab. If the pull down menu item has a small solid triangle at the end, it has a *flyout* menu associated with it. If you move the cursor on the triangle, the flyout menu will display. See Figure 2-2.

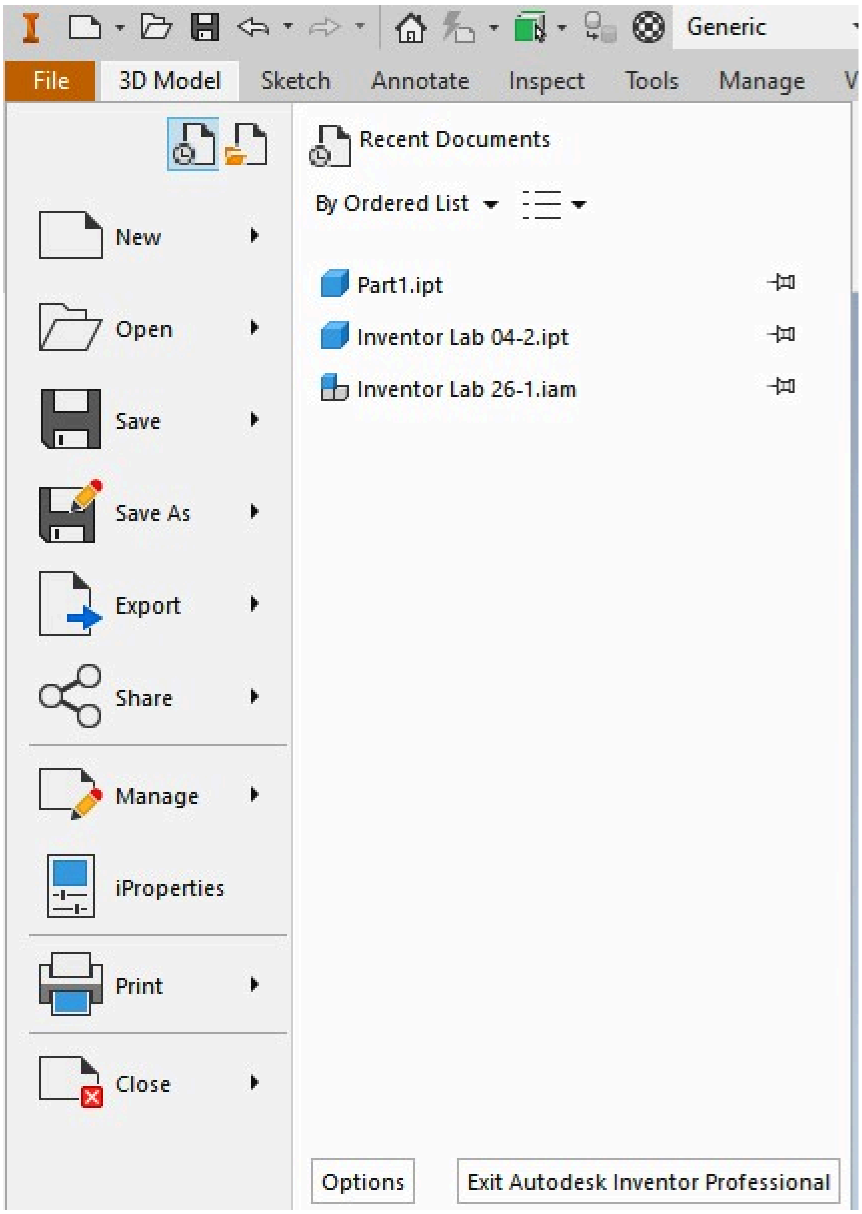


Figure 2-2
The Pull-Down Menu

Ribbon Menu

The Ribbon menu is used for most of your work in Inventor. See Figure 2-3.

Figure 2-3
A Typical Ribbon Menu

Browser Bar

The *Browser bar* displays the hierarchical structure of the model or assembly of the current file. It is your most important and most-used tool to create and modify objects within files. The Browser bar will be taught in more detail in future modules. It is normally docked on the left side of the Graphic window. See Figure 2-4.

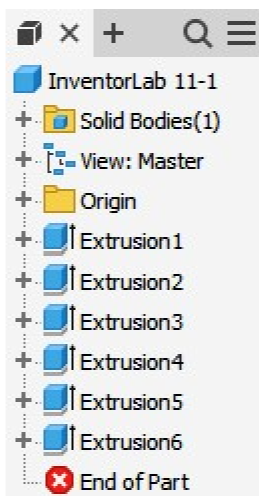


Figure 2-4
Browser Bar

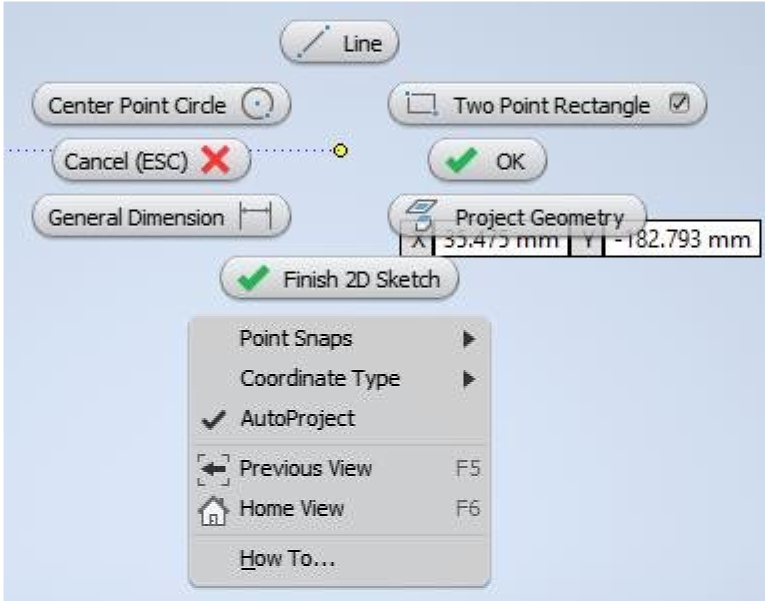


Figure 2-5 Right-click Menu

Right-Click Menu

When the right mouse button is clicked, it displays the *Right-click menu*. See Figure 2-5. It is sometimes called the *Cursor menu* since it displays at the current location of the cursor. This menu changes automatically depending on the current command or operation being performed. It usually takes the new user a bit of time to get used to using this menu.

Status Bar

The *Status bar* is permanently located along the bottom of the Graphic window. It displays the prompts that the current command is issuing, as shown in Figure 2-6, as well as the coordinate locations, as shown in Figure 2-7. These prompts help you understand what information Inventor requires and the cursor location, length, and angle in the current command.

Figure 2-6 Status Bar – Showing a Command Prompt.

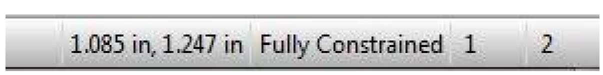


Figure 2-7 Status Bar – Showing Coordinate Information

The Graphic Cursor

The *Graphic cursor* is used to select menu items or objects on the sketch or model. See Figure 2-8.

Figure 2-8
The Graphic Cursor

Dialogue Boxes

Inventor commands use many different dialogue boxes to obtain information to be used by the command or current operation. A typical dialogue box is shown in Figure 2-9.

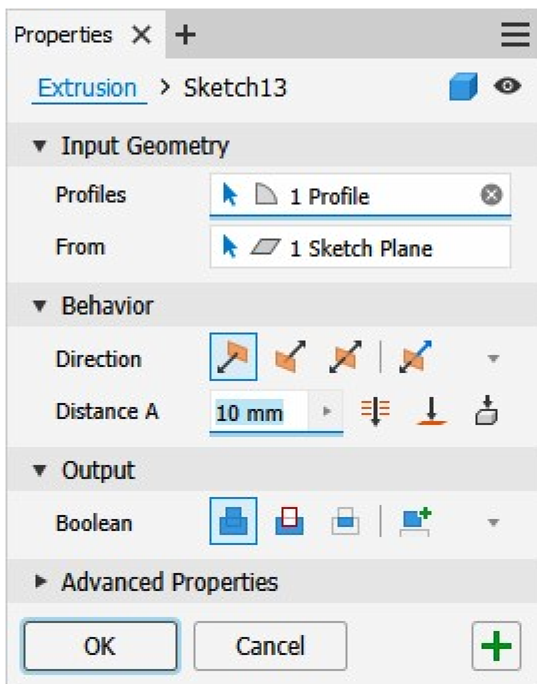


Figure 2-9 A Typical Dialogue Box

The Mouse

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

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

Middle Button: The middle button or the wheel is used to zoom and pan around the Graphic window. This will be discussed in detail in Module 3.

Right Button: The right button displays the Right-click menu. See Figure 2-10. The Right-click menu will change depending on the current command or operation being preformed. It is a very helpful menu and should be used as often as possible when working in Inventor.

Inventor Commands

An Inventor *command* is an instruction from you to Inventor instructing it what operation to perform. Commands can be entered by selecting an item from a Pull-down menu, a Right-click menu, an icon on a Toolbar, an item on the Panel menu, or entering a shortcut on the keyboard. Since there are usually many different ways of entering the same command, you should select the method that works the best for you. There is no right or wrong way to enter a command. You should experiment to find the fastest method to improve your drawing speed and productivity.

Ending the Current Command

When you enter a command, it becomes the *current command* or sometimes called the *active command*. Inventor must be instructed to end the current command. There are two methods available to do this. The first is to press the Esc key on the keyboard and the second is click to Done or Cancel on the Right-click menu as shown in Figure 2-10.

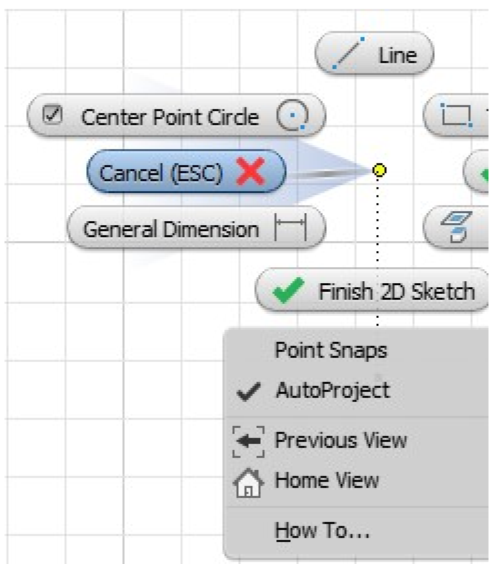


Figure 2-10 Ending a Command

Undo and Redo Commands

In the current file, any Inventor command or commands can be undone to reverse any changes that they may have made. To do this, click the Undo icon in the Quick Access toolbar as shown in Figure 2-11. Each click will undo the previous step. If the Undo icon was used to undo a command or a series of commands, it can be reversed by clicking the Redo icon. See Figure 2-12.

Fig 2-11 Undo Icon

Fig 2-11 Redo Icon

WORK ALONG: Creating and Saving a Part File

Step 1

Start Inventor and check the current project. If required, set it to Inventor Course. (Figure Step 1)

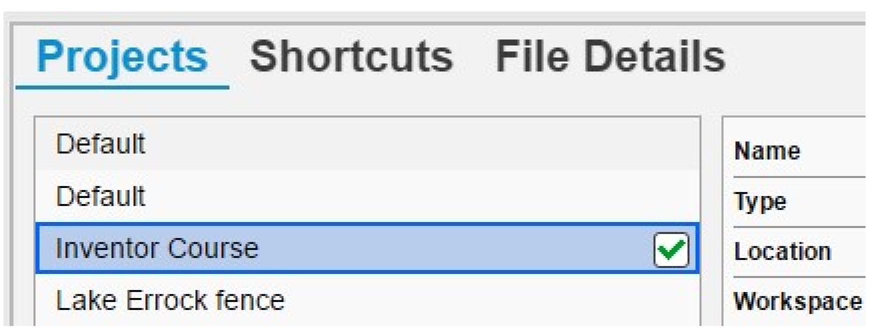


Figure Step 1

Step 2

Click the NEW command icon in the upper left corner of the screen or under the File tab. In the Create New File dialogue box, enable the folder: Templates – English. (Figure Step 2A and 2B)

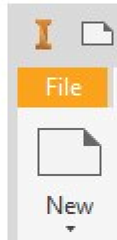


Figure Step 2A

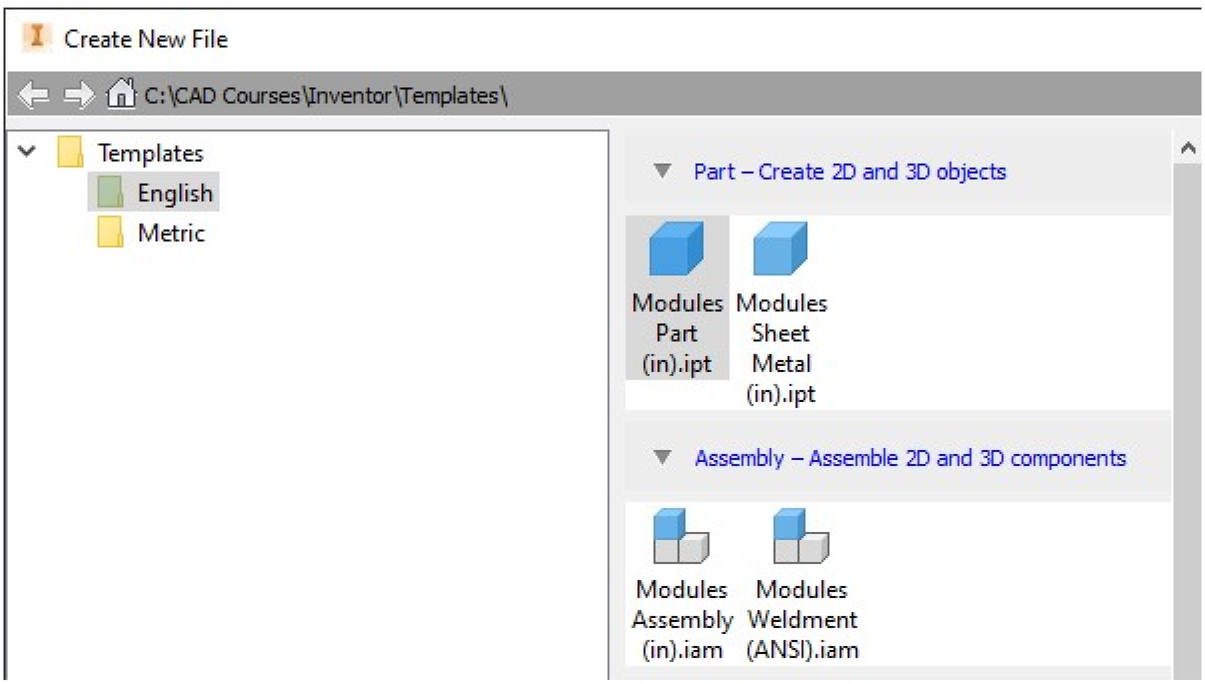


Figure Step 2B

AUTHOR'S COMMENTS: If the Create New File dialogue box does not display template files where all the names begin with Modules (i.e. Modules Part (in).ipt), stop this workalong and go back and redo Module 1.

Step 3

Select the template: Modules Part (in).ipt icon and click OK. (Figure Step 3)

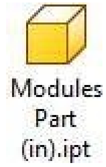


Figure Step 3

Step 4

Inventor will display the Graphic window, in Sketch mode, as shown in the figure. (Figure Step 4)

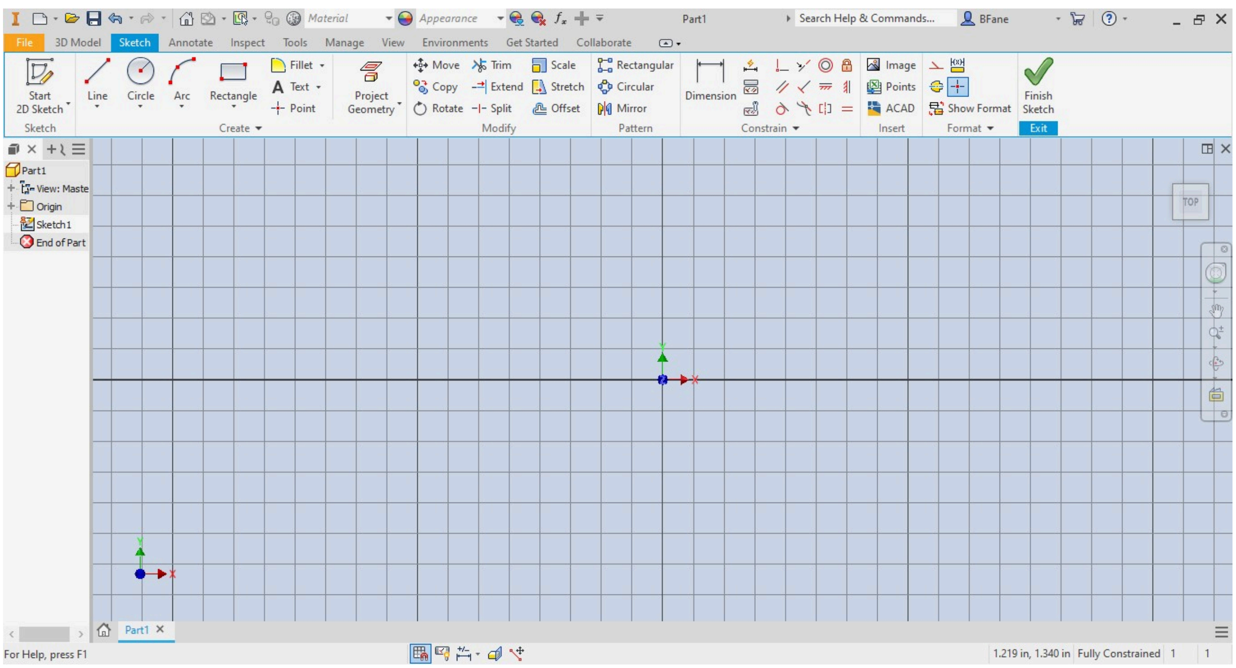


Figure Step 4 [Click to see image full size]

Step 5

Click the Finish Sketch icon to change to Model mode. (Figure Step 5A and 5B)

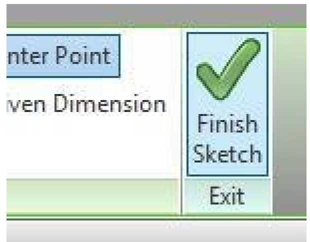


Figure Step 5A

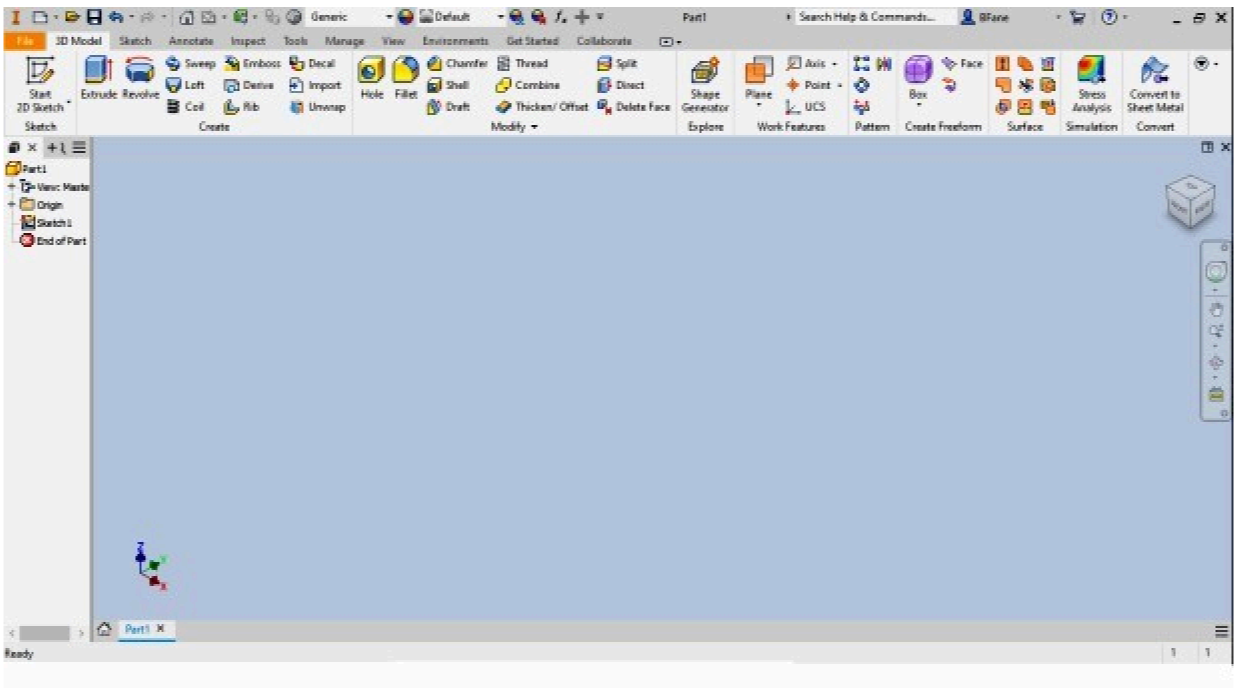


Figure Step 5B [Click to see image full size]

Step 6

Click the Save icon. It will open the Save As dialogue box. In the File name: box, enter the name: Inventor Workalong 02-1 and click Save. (Figure Step 6A and 6B)

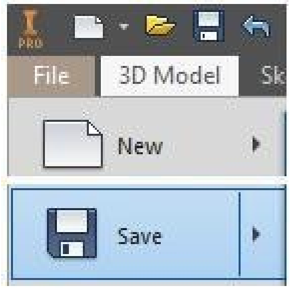


Figure Step 6A

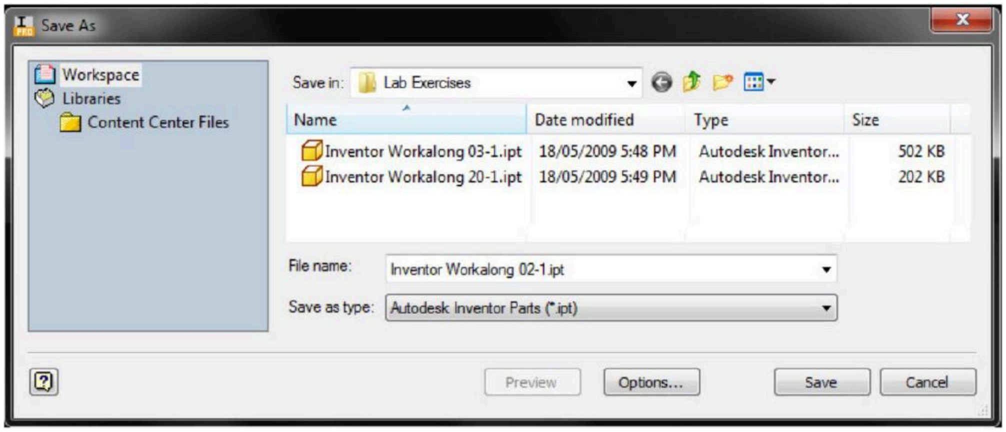


Figure Step 6B [Click to see image full size]

AUTHOR'S COMMENTS: Ensure that the Save in: box displays the folder Lab Exercises. If it does not, check to ensure that the current project is set to Inventor Course

Step 7

The file name should now display on top bar of the Graphic window similar to what is shown in the figure. (Figure Step 7)

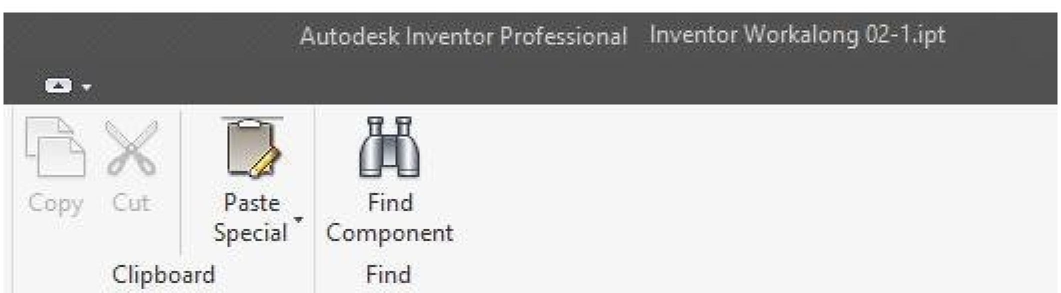


Figure Step 7

Step 8

Click the *File* tab and then the *Close* icon to close the current part file. (Figure Step 8)

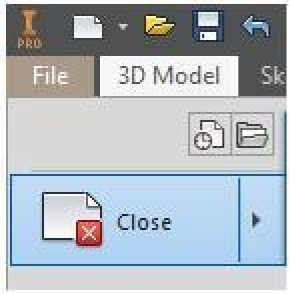


Figure Step 8

Step 9

Click the Inventor icon to pull down the menu. On the Pull-down menu, click *Exit Autodesk Inventor Professional* to exit Inventor. (Figure Step 9)

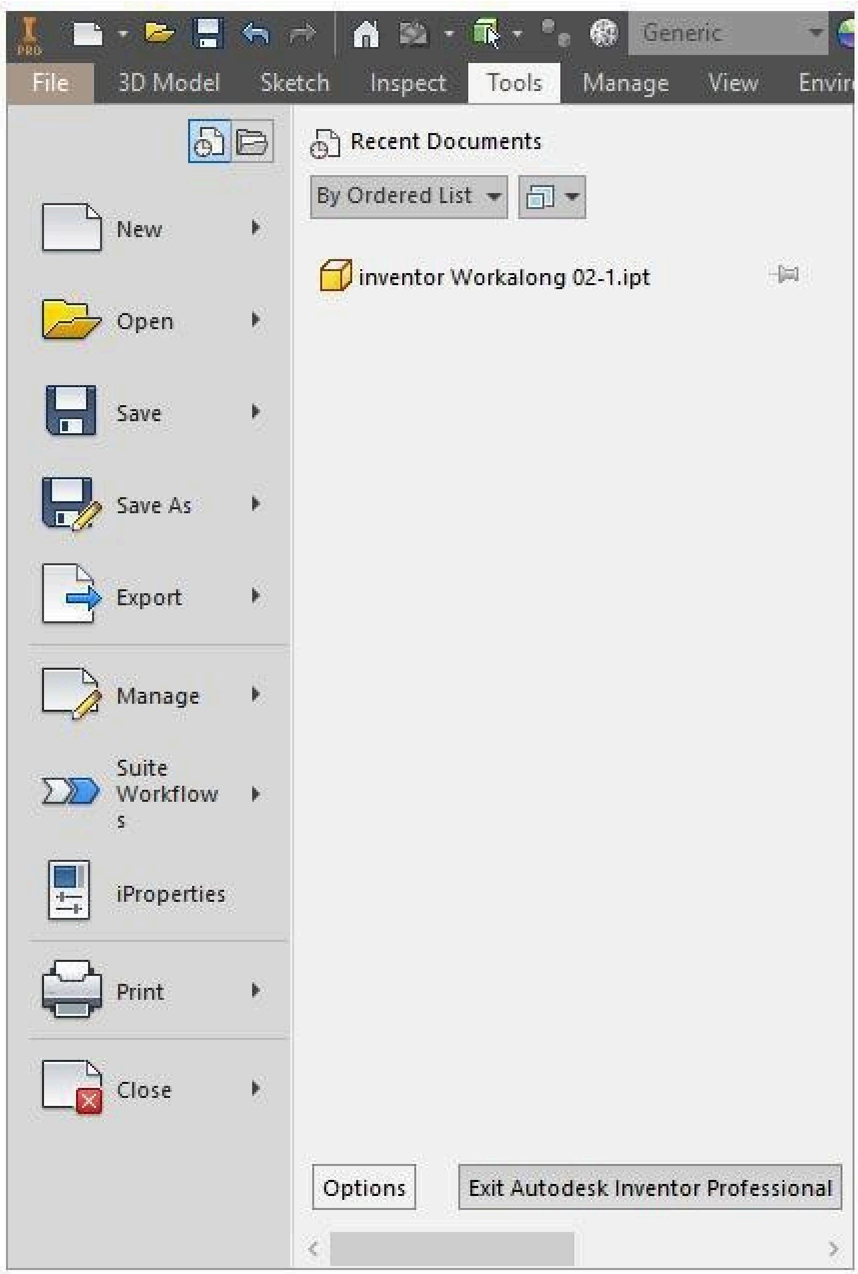


Figure Step 9

WORK ALONG: Configuring and Working with Inventor's Interface

Step 1

Start Inventor and check the current project. If required, set it to Inventor Course.

Step 2

Click the New icon. In the New File dialogue box, enable the folder: Metric Templates.

Step 3

Select the template: Module Part (mm).ipt. (Figure Step 3)

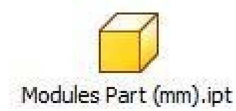


Figure Step 3

> **AUTHOR'S COMMENTS:** The units for this part are millimeters.

Step 4

Click Finish Sketch to exit Sketch mode. While in Model mode, save the part file with the name: Inventor Workalong 02-2. (Figure Step 4)

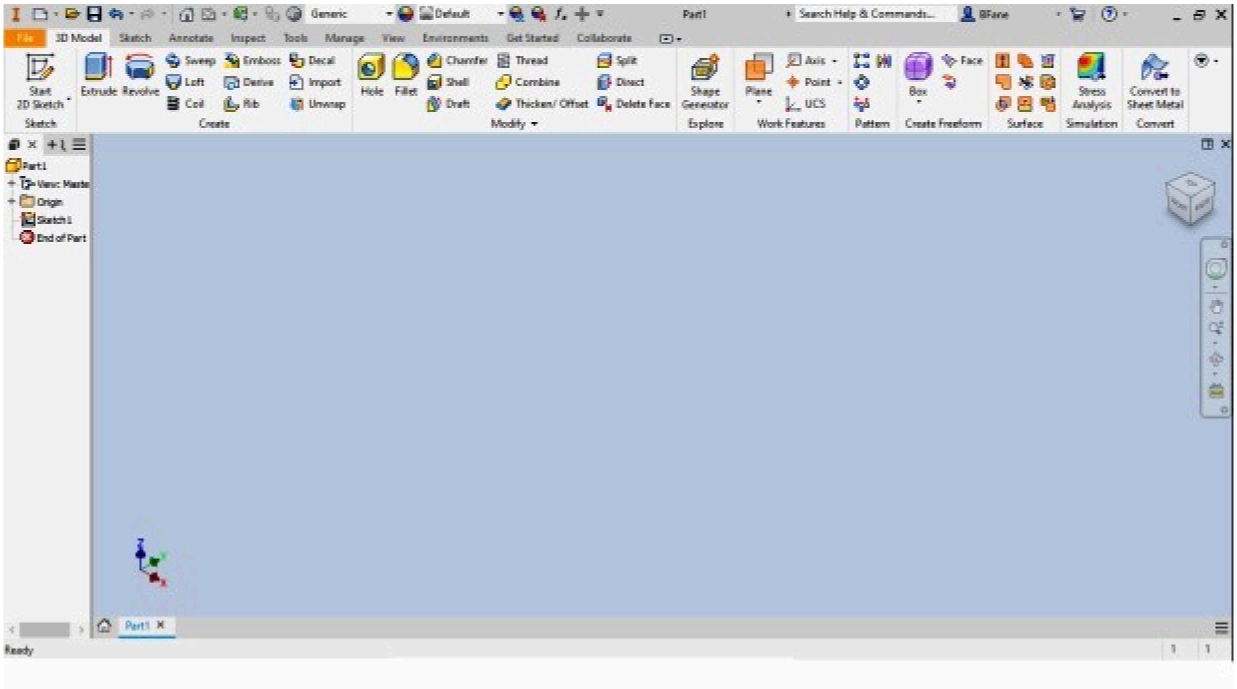


Figure Step 4 [Click to see image full size]

AUTHOR'S COMMENTS: You must be in Model mode to save the current part file.

Step 5

Move the cursor onto the space between the plus (+) sign and the magnifying glass at the top of the Browser bar and press and hold the left mouse button down. While holding the button down, drag the Browser bar into the Graphic window. The Graphic window should appear similar to the figure. (Figure Step 5)

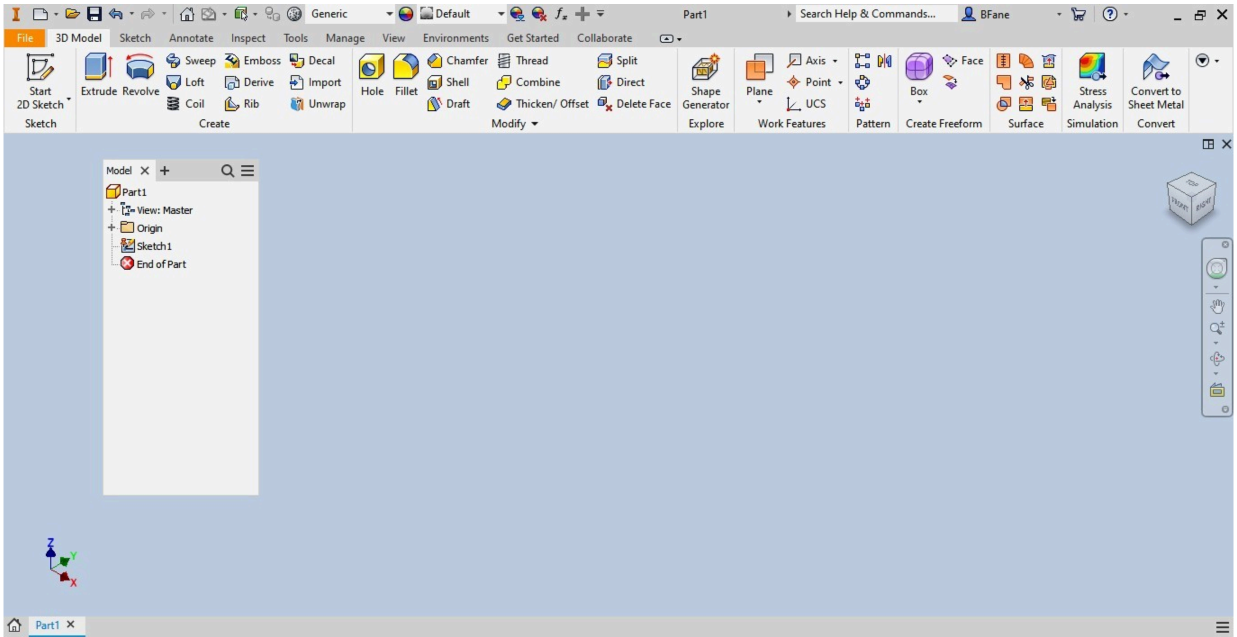


Figure Step 5 [Click to see image full size]

AUTHOR'S COMMENTS: I had you do Step 5 to show you how to undock and move the Browser bar onto the Graphic window. When you do this, it is called a floating menu. When it is attached to one of the four sides, it is called a docked menu.

USER TIP: The Status bar displays Inventor prompts to the you. Watch it closely as it will display what information the current command is looking for from you or coordinate information of the current cursor location.

Step 6

Move the cursor onto the space between the plus (+) sign and the magnifying glass at the top of the Browser bar and press and hold the left mouse button down. While holding the button down, drag the Browser bar to the left until its colors fade. Release the left mouse button and the browser bar will dock on the left edge of the screen. (Figure Step 6)

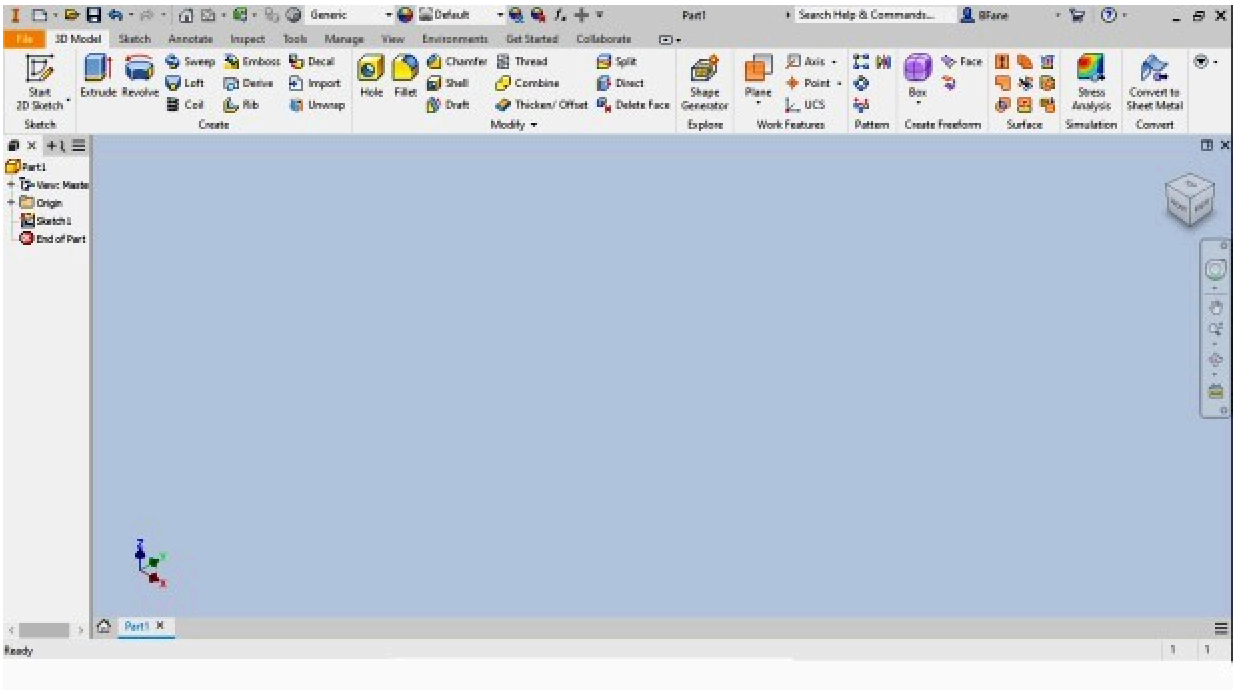
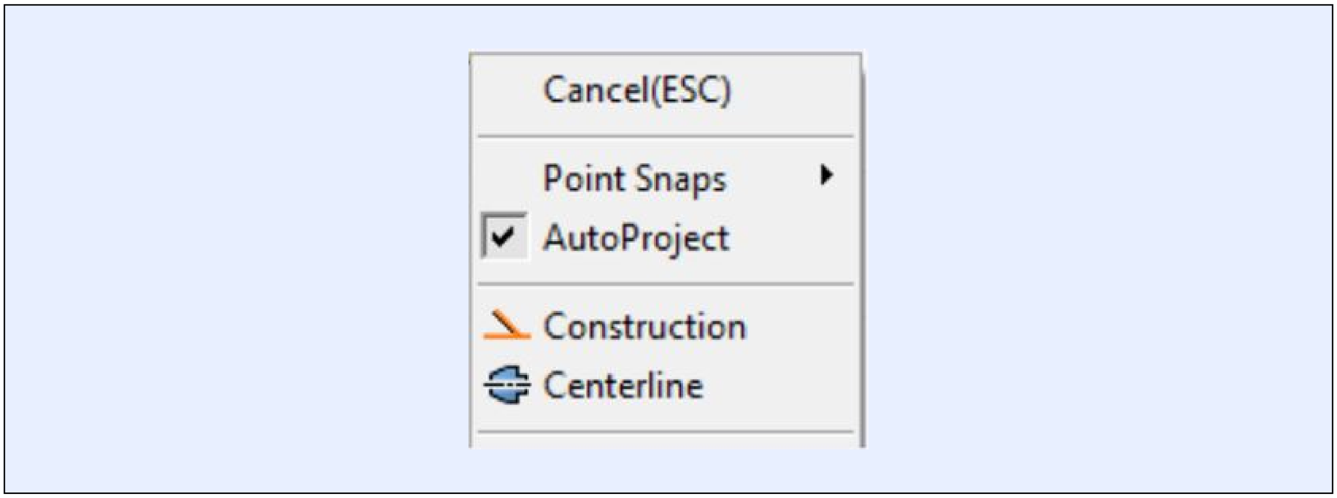


Figure Step 6 [Click to see image full size]

AUTHOR'S COMMENTS: You may have to practice docking and undocking the menu and moving them around the Graphic window until you get comfortable doing it. The Graphic window configuration shown in Figure Step 6 is the one I use when working in Inventor. I suggest thatyou use it while working on the workalongs and lab exercises in the Inventor book.

MUST KNOW: Either the Esc key on the keyboard or the Cancel on the Right-click menu must be used to end the current command. You must always let Inventor know when to terminate the current command.



Step 7

Click Sketch1 in the Browser bar to select it and right click the mouse. In the Right-click menu, click Edit Sketch. (Figure Step 7)

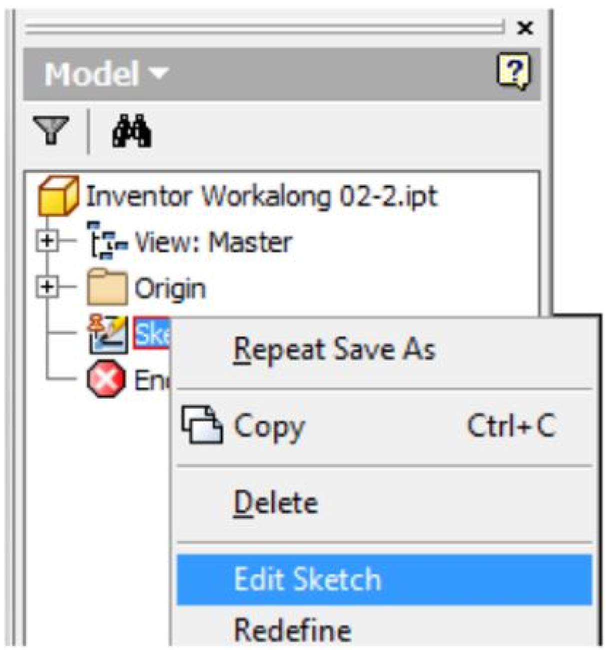


Figure Step 7

Step 8

The Graphic window will change to Sketch mode and should appear similar to the figure. (Figure step 8)

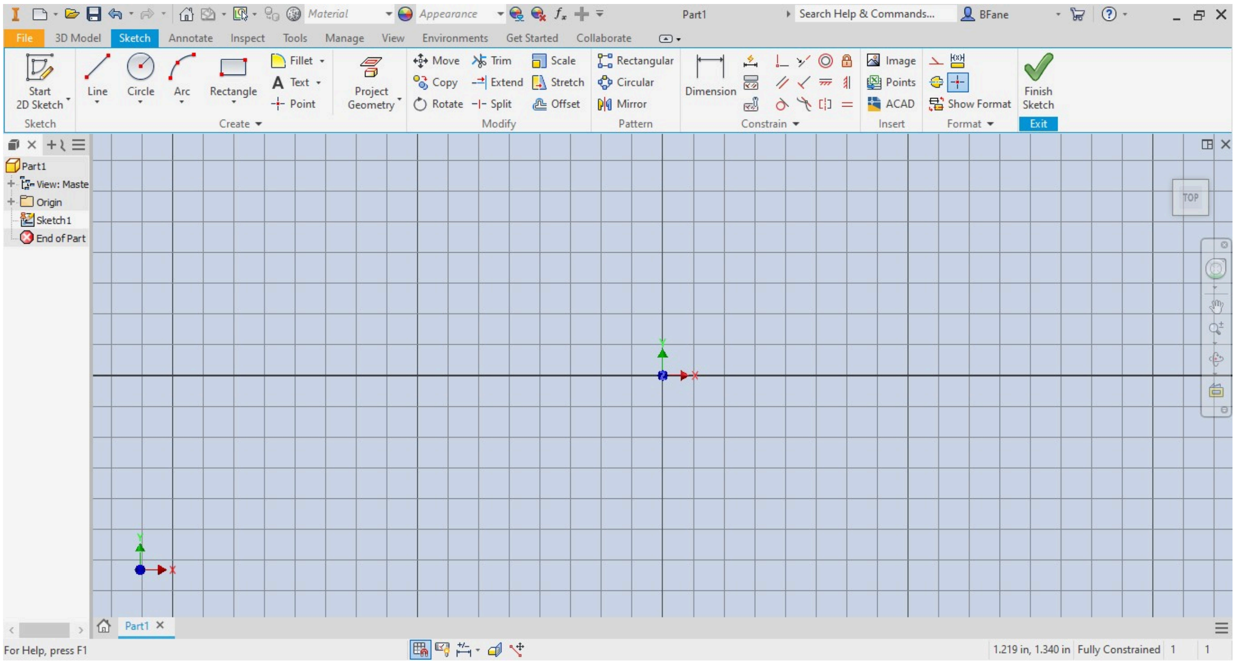
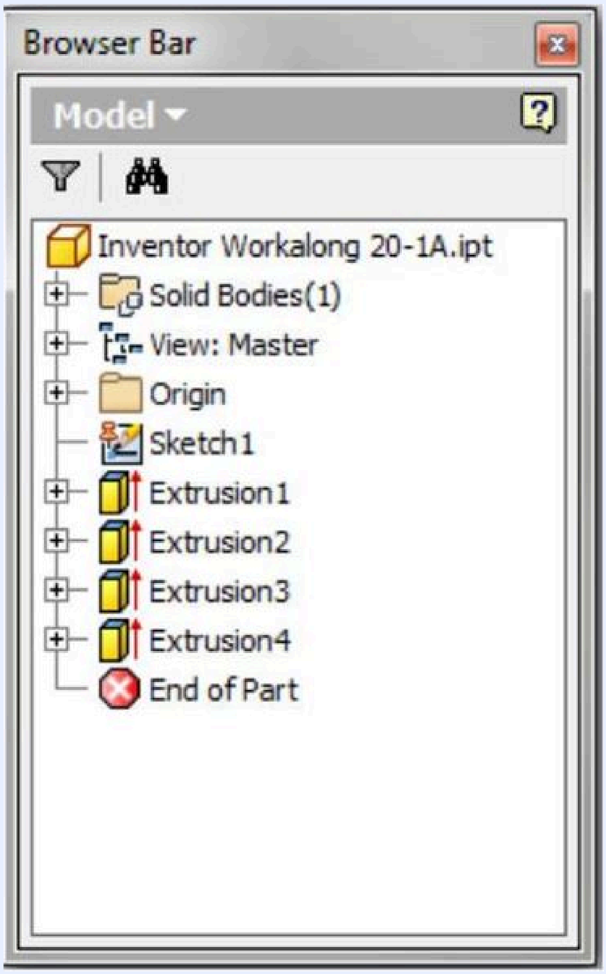


Figure Step 8 [Click to see image full size]

Step 9

Change to Model mode. Save and close the part file.

MUST KNOW: The Browser bar displays the hierarchical structure of the model or assembly of the current file. It is your most important and most-used tool to create and modify files.



MUST KNOW: A part file is one 3D solid model. A part file has the file extension .ipt. The extension .ipt is an acronym for Inventor ParT. Working drawings can be created from the part file or part of an assembly or presentation file.

Key Principles

Key Principles in Module 2

1. When a file is active, Inventor will display the Graphic window. The Graphic window has two different modes, the Sketch mode and the Model mode. They are used to create 3D solid models, called
2. A part file is one 3D solid model. A part file has the file extension .ipt. The file extension .ipt is an acronym for Inventor ParT. Working drawings can be created from the part file or part of an

assembly or presentation file.

3. Every new Inventor file must be created from an existing template
4. Inventor keeps the current file in RAM memory. If the computer cashes or the power fails, all the work on the file, since the last time it was saved, will be lost. When the file is saved, Inventor saves what is in RAM memory onto the disk drive. Ensure that the current file is saved frequently to avoid losing production Saving it every 5 to 10 minutes is a good habit to get into.
5. The Browser bar displays current file's hierarchical
6. The Status bar, located across the bottom of the Graphic window, is a very important part of the operator's day-to-day work in Inventor.
7. Inventor must be instructed to end the current command. There are two methods available to do this. The first is to press the Esc key on the keyboard and the second is to click Cancel or Done on the right-click menu.

Module 3 Viewing the 2D Sketch and 3D Model

Learning Outcomes

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

1. Describe how to change the viewing position of the 2D sketch and the 3D model by zooming, panning, and orbiting.
2. Apply the OPEN command to open existing Inventor files.
3. Apply the ZOOM, ORBIT, ZOOM ALL, PAN, and HOME VIEW commands or use the wheel on the mouse to change the viewing position of the 2D sketch and 3D model.

Viewing the 2D Sketch and 3D Model

In Sketch mode, you are viewing and working on a two dimensional plane. In Model mode, you are viewing and working in three dimensions the same way the human eye see real objects. It is essential for you to be able to change the viewing position of the 2D sketch and the 3D model as required. This is done by zooming, panning, and orbiting to change the viewpoint of the 2D sketch and the 3D model.

Zooming

Zooming is the process of changing the viewable size of the sketch or model to make it appear either smaller or larger. It is an important tool for you and is used extensively in the drawing and modeling process. As it is zoomed, the size of the sketch or model not changed, Inventor is simply adjusting the distance the object is from your eyes making it appear larger or smaller. I wondered why the Frisbee was getting bigger, and then it struck me…

Panning

Panning is the process of moving the sketch or model around the Graphic window without actually physically moving it from its location in space. It is an important tool for you and is used extensively while working in Inventor.

Orbiting

Orbiting is the process of changing the orientation of the sketch or model in relation to your eyes. Rather then changing the orientation of your eyes, Inventor orbits the sketch or model and your eyes remain stationary. The model and sketch are not physically rotated or their orientation in space changed. Orbiting is used extensively when working in Inventor.

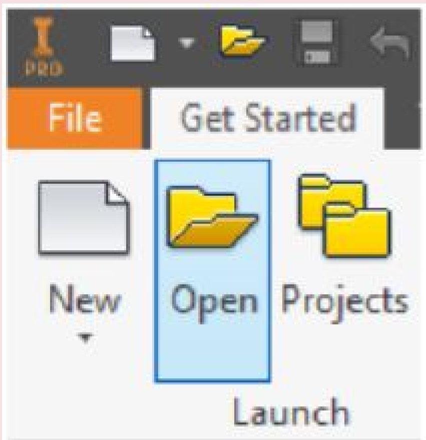
Home View

The *Home view* is important to you while working in Inventor. It is the viewing position of the sketch or model to a known isometric view. This view re-establishes your bearings to better visualize the sketch or model because you are viewing it in a known viewing position.

Inventor Command: OPEN

The OPEN button under the *Get Started* tab command is used to open an existing Inventor file.

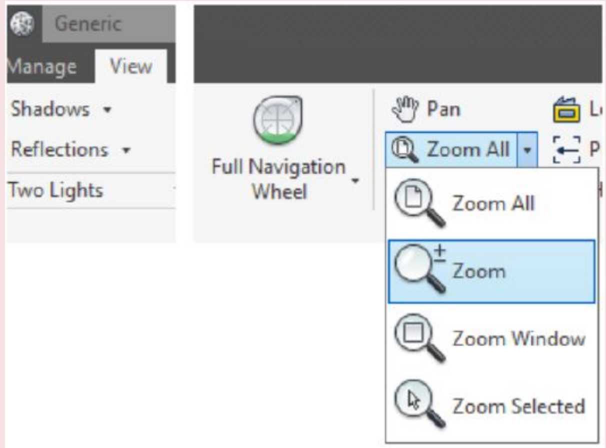
Shortcut: **CTRL+O**



Inventor Command: ZOOM

The ZOOM command is used to zoom the sketch or model.

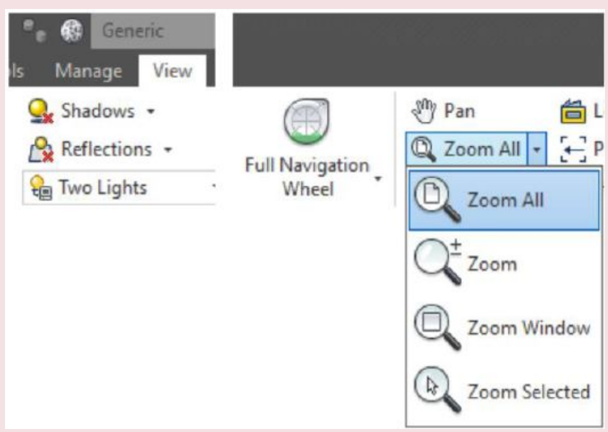
Shortcut: **F3**



Inventor Command: ZOOM ALL

The ZOOM ALL command is used display the current sketch or model to fit inside the Graphic window.

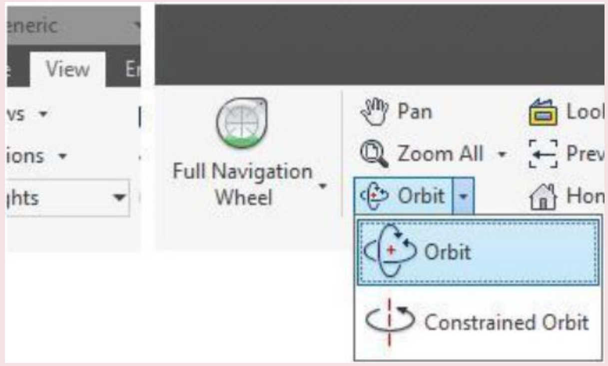
Shortcut: **HOME**



Inventor Command: ORBIT

The ORBIT command is used to orbit the sketch or model around the X, Y, or Z axes. The sketch or model can be orbited while another command is active.

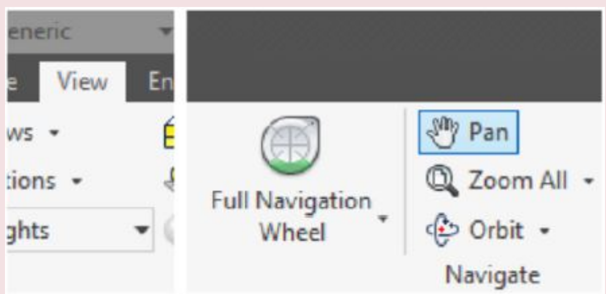
Shortcut: **F4**



Inventor Command:PAN

The PAN command is used to move the sketch or model in the Graphic window.

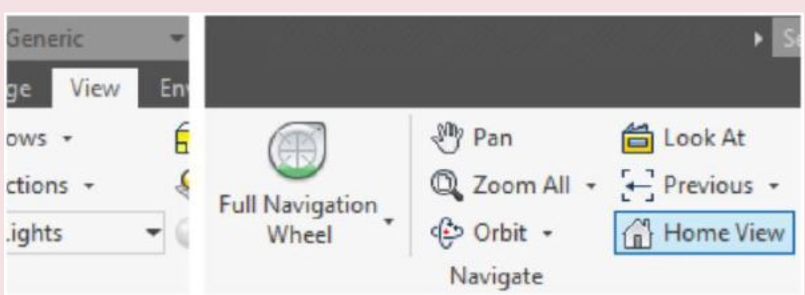
Shortcut: **F2**



Inventor Command: HOME VIEW

The HOME VIEW command is used to display the sketch or model in the home view.

Shortcut: **F6**



WORK ALONG: Viewing the Model

Step 1

Start Inventor and check the current project. If required, set it to Inventor Course.

Step 2

Click the Open icon. In the Open dialogue box, select the file:

Inventor Workalong 03-1.ipt . Note how the file name will appear in the File name: box as shown in the figure. (Figure Step 2)

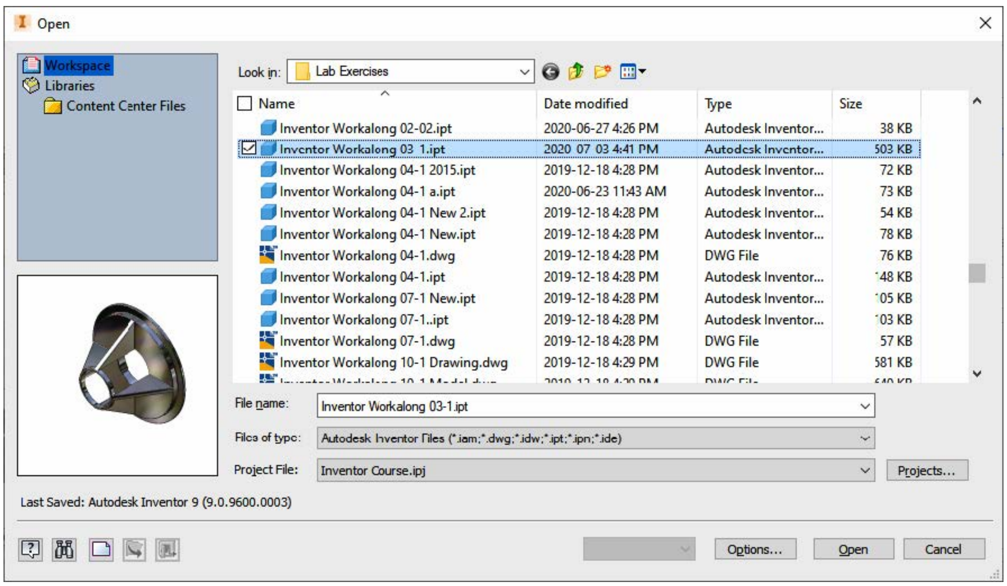


Figure Step 2 [Click to see image full size]

Step 3

If you are asked to convert the appearance, click Yes. (Figure Step 3).

Figure Step 3

Step 4

The part will open and display in Model mode and appear similar to the figure. (Figure Step 4).

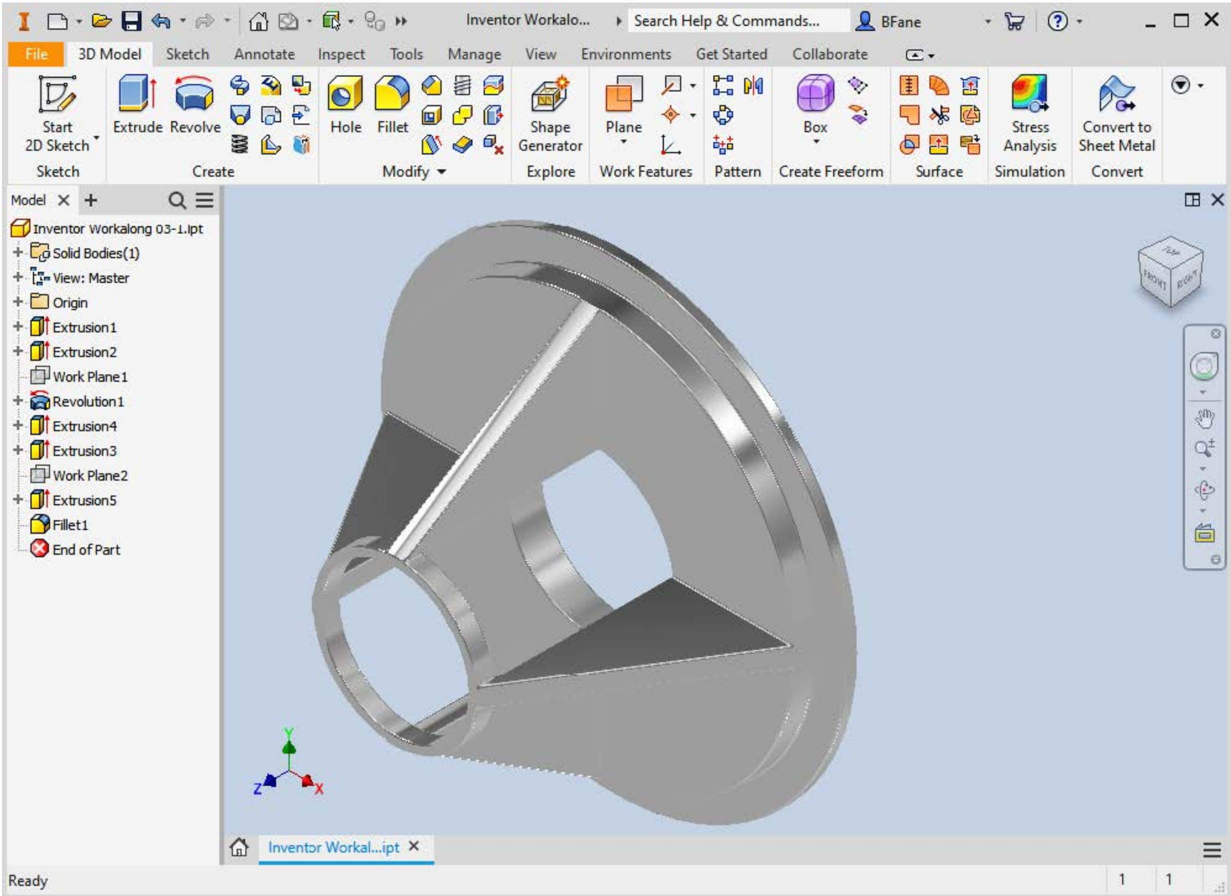


Figure Step 4 [Click to see image full size]

AUTHOR'S COMMENTS: This is a 3D solid model that was created in Inventor and supplied with the Inventor book.

AUTHOR'S COMMENTS: For this workalong, I am assuming that you have a mouse with a wheel as the middle button. If you do not have a wheel on your mouse, I strongly suggest that you change it with a mouse that has one. It will save you a great deal of drawing time.

Step 5

Move the cursor to approximately the centre of the solid model as shown in the figure. While keeping the cursor in the centre, rotate the mouse wheel back and forth. Note how the model appears larger and smaller. (Figure Step 5A and 5B)

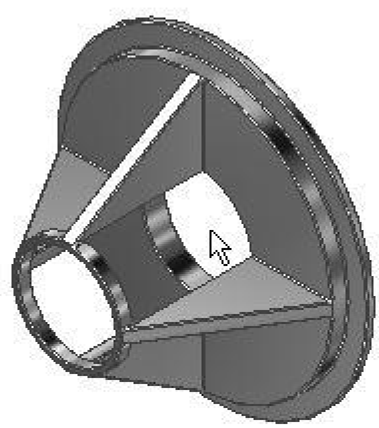


Figure Step 5A

AUTHOR'S COMMENTS: Inventor will zoom the model using the location of the cursor as the centre point of the zoom. That is why it is important to locate the cursor in the centre of the model when zooming.

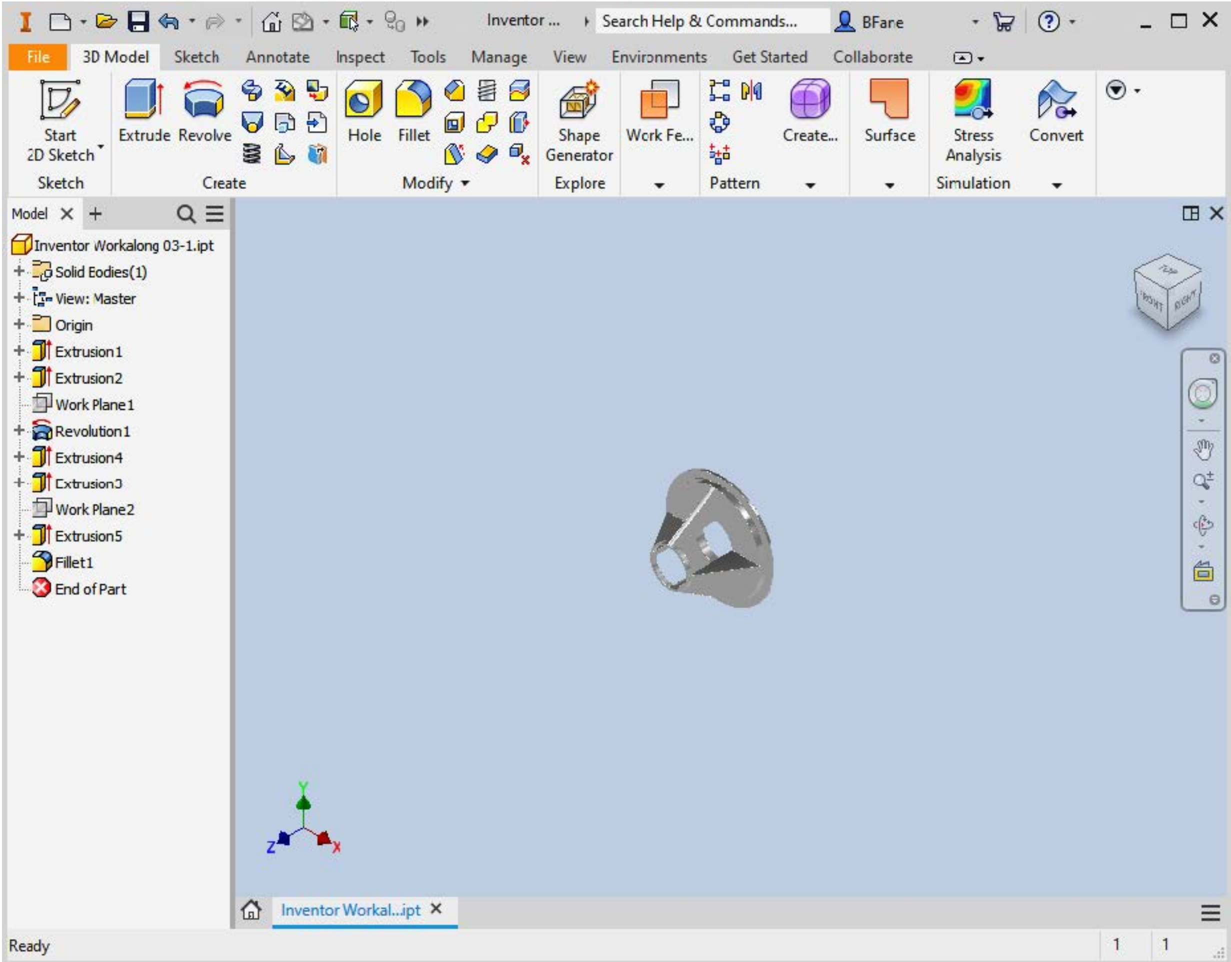
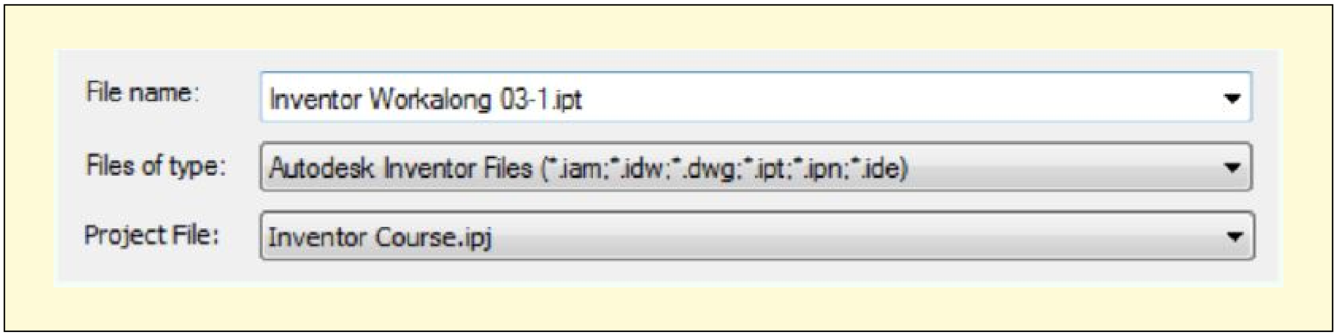


Figure Step 5B [Click to see image full size]

AUTHOR'S COMMENTS: If you don't have a wheel on your mouse, press F3 and hold it down. While holding it down, press the left mouse button and move the mouse forward to achieve the same results as Step 5.

USER TIP: When opening a file in Inventor, ensure that the Open File dialogue box is listing the correct file type(s). To list part files which have the extension .ipt , ensure that *.ipt file type is listed in the File of type: box as shown below. The * means that all files that have the extension .ipt.


File name:
Inventor Workalong 03-1.ipt
Files of type:
Autodesk Inventor Files (*.iam;*.idw;*.dwg;*.ipt;*.ipn;*.ide)
Project File:
Inventor Course.ipj

Step 6

Move the cursor near the model and press and hold the wheel down. A small Hand icon will replace the arrow cursor as shown in the figures. While the Hand icon is displayed, move the mouse to pan the model as shown. (Figure Step 6A, 6B, and 6C)

> **AUTHOR'S COMMENTS:** If you don't have a wheel on your mouse, press F2 and hold it down. While holding it down, press the left mouse button and move the mouse to achieve the same results as Step 6.

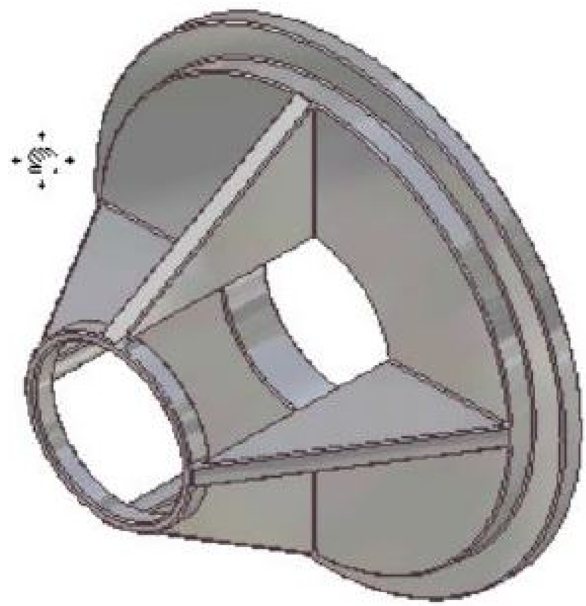


Figure Step 6A

Figure Step 6B

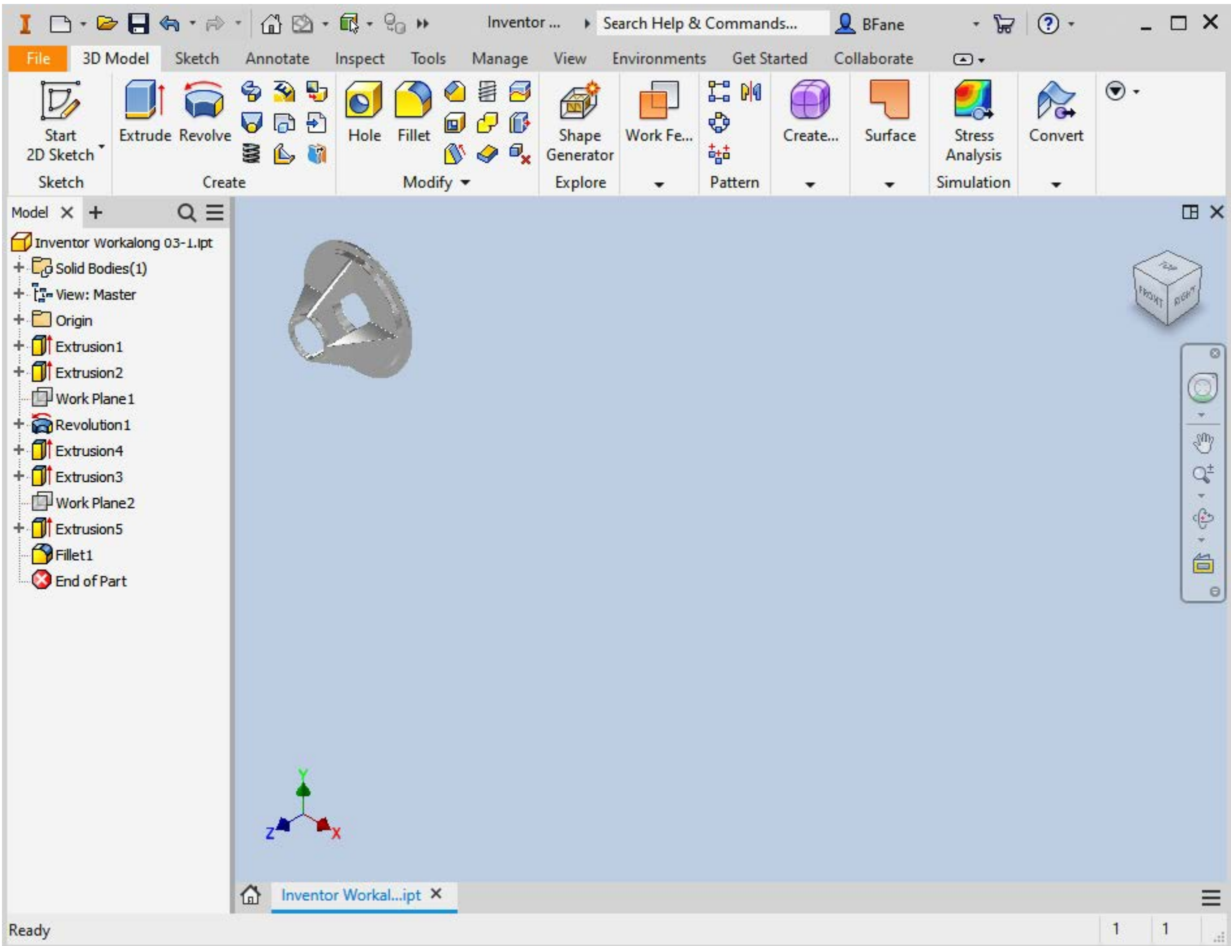


Figure Step 6C [Click to see image full size]

MUST KNOW: The HOME VIEW command (F6) is a very important command. It is used to change the viewing position of the 2D sketch or 3D model to a known isometric view. By doing this, it re-establishes your bearings to a known viewing position.

Step 7

Press F6 to return the model to its Home view. (Figure Step 7)

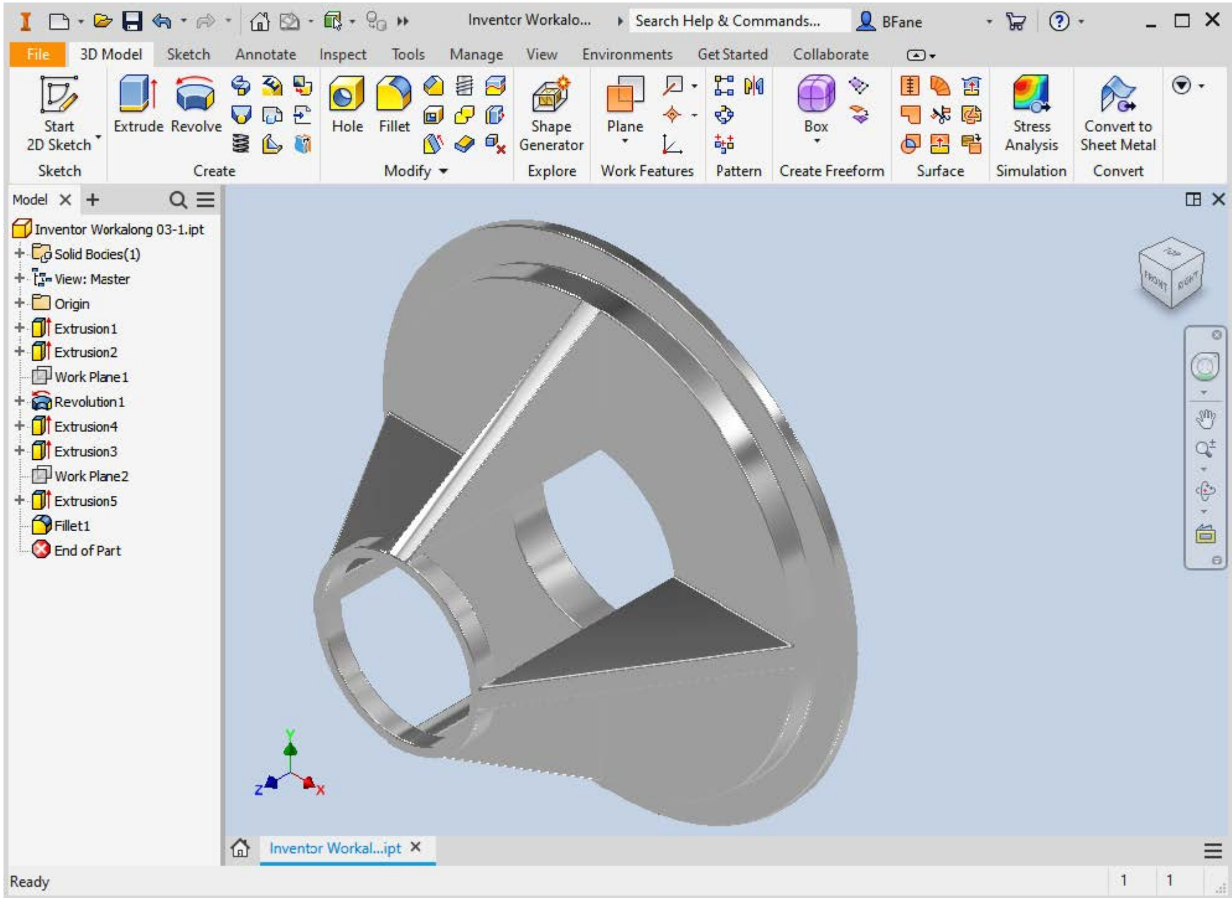


Figure Step 7 [Click to see image full size]

AUTHOR'S COMMENTS: The Home View command (F6) is a very important command. It is used to change the viewing position of the 2D Sketch or 3D model to a known isometric view. You can re- establish your bearings to better visualize the sketch or model.

Step 8

Press F4 and hold it down. While holding it down, move the cursor outside the orbit circle, as shown below. The cursor will change as shown in the figures. Press and hold down the left mouse button. While holding it down, move the mouse and the model will orbit. (Figures 8A, 8B, and 8C)

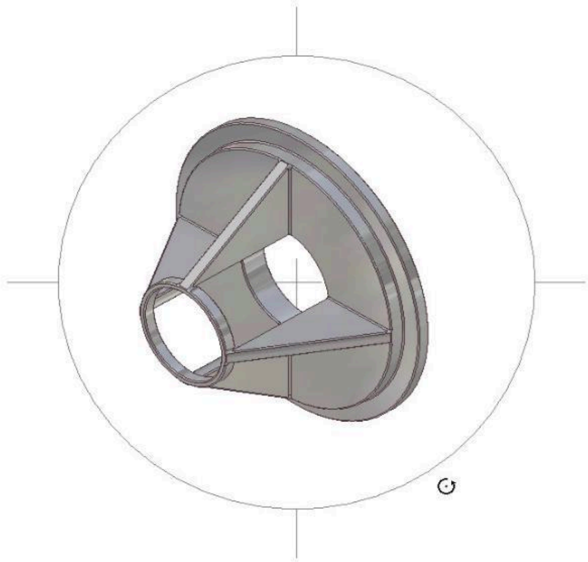


Figure Step 8A

Figure Step 8B

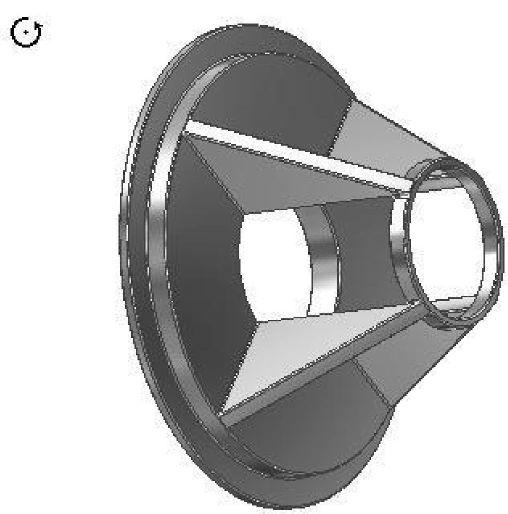


Figure Step 8C

AUTHOR'S COMMENTS: When the cursor appears as shown in Figure Step 8B, you are rotating the model around the Z axis only.

Step 9

Press F4 and hold it down. While holding it down, move the Graphic cursor somewhere inside the orbit circle. The icon will change in appearance as shown in the figures. Press the left mouse button and while holding it down, move the mouse. The model will orbit. (Figure Step 9A, 9B, and 9C)

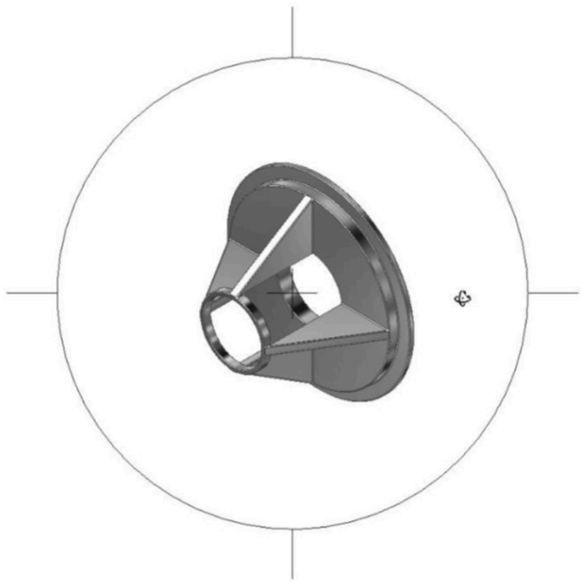


Figure Step 9A

Figure Step 9B

Figure Step 9C

AUTHOR'S COMMENTS: When the cursor displays as shown in Figure Step 9B, you are orbiting the model around all three axises, X, Y and Z at the same time.

USER TIP: Many Inventor commands have shortcut keys to speed the drawing process. For example, the F4 is the shortcut for the ORBIT command. Using shortcuts are faster than using a menu. It is best to learn and use the shortcuts whenever possible to shorten the drawing time. Inventor shortcuts are shown in the Inventor book and on tooltips in the Inventor menus.

Step 10

Press F6 to return the model to its home view. (Figure Step 10)

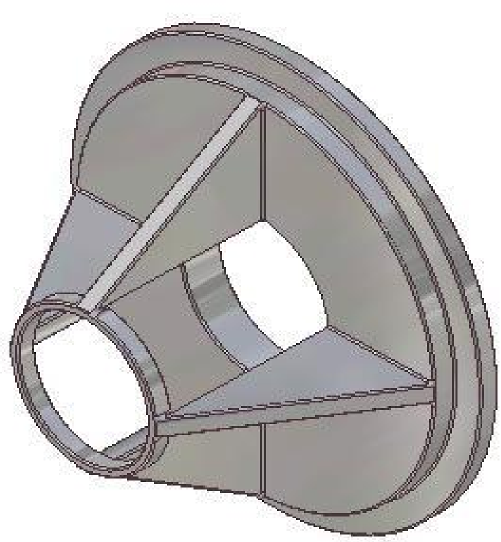


Figure Step 10

AUTHOR'S COMMENTS: Now that you know how to zoom, pan, and orbit the 3D model. Practice doing this until you are comfortable changing your viewpoint of the model.

Step 11

Close the part without saving it.

USER TIP: While it is not absolutely necessary, it is very helpful to have a mouse with a wheel as the middle button. Rather then using the ZOOM and PAN commands, you can zoom and pan the sketch or model using the wheel. Using the wheel rather then the commands to zoom and pan, will greatly improve your drawing productivity.

Key Principles

The Key Principles in Module 3

1. It is essential for you to be able to change the viewing position of the 2D sketch and 3D This is done by zooming, panning, and orbiting to change viewpoint of the sketch or model.
2. The HOME VIEW command is a very important command. Use it to change the viewing position of the 2D sketch or 3D model to a known isometric view. You can re-establish your bearings to better visualize the sketch or model.
3. Use shortcut keys whenever possible to speed your production. Shortcut keys taught in this module are Home – Zoom All, F2 – Pan, F3 – Zoom, F4 – Orbit, and F6 – Home View.

Module 4 Sketching Lines

Learning Outcomes

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

1. Describe the Cartesian Coordinate System, parametric solid modeling, the Base sketch, and geometrical constraints.
2. Describe snapping onto grids, lines, endpoints, and midpoints.
3. Describe and apply the PROJECT GEOMETRY command to project the Center Point onto the Base sketch.
4. Following Inventors 2D sketching rules, describe and apply the LINE command to draw the Base sketch of simple solid models.

Geometry Lesson: Points and Lines

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

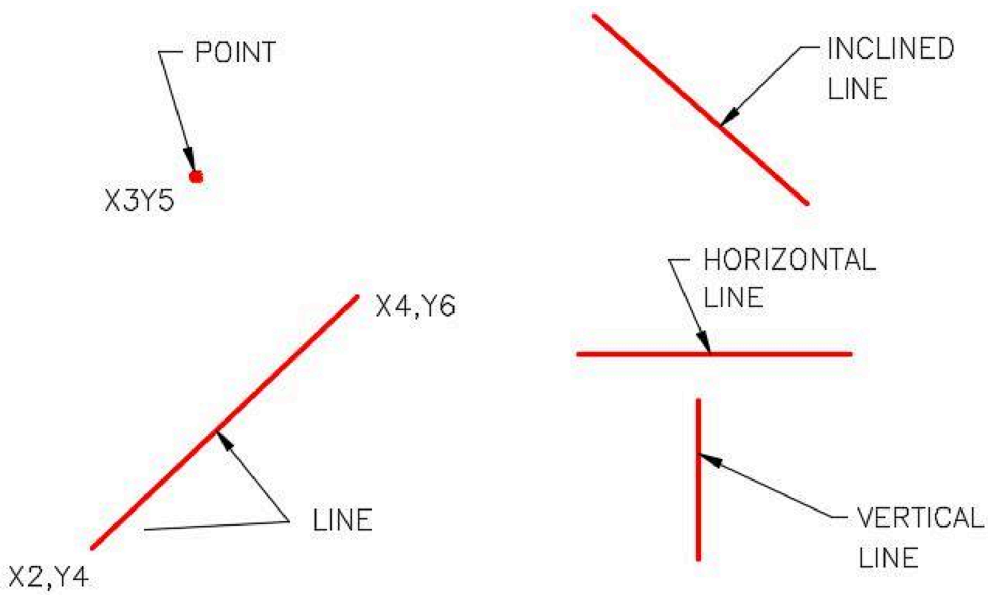


Figure 4-1
Points and Lines [Click to see image full size]

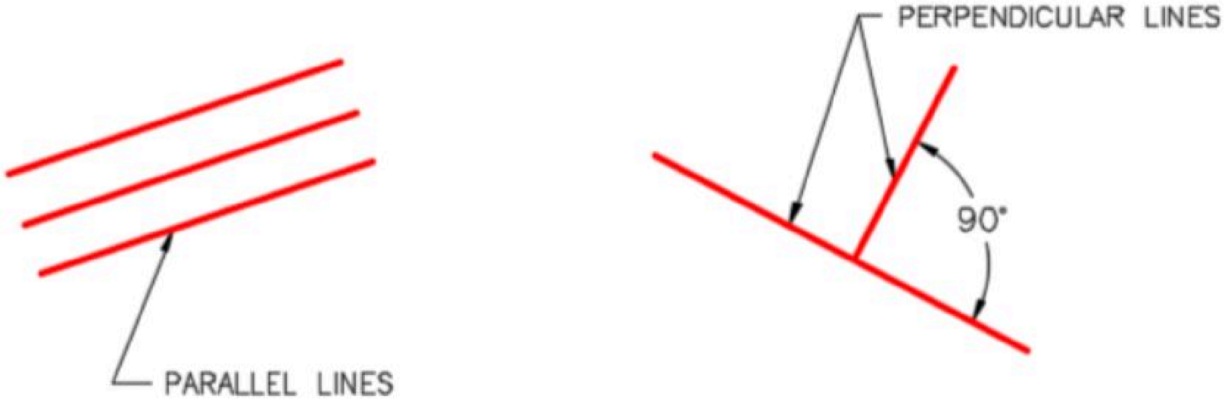


Figure 4-2
Parallel and Perpendicular Lines [Click to see image full size]

The Cartesian Coordinate System

To accurately draw two dimensional (2D) Base sketches, you must understand the Cartesian Coordinate System.

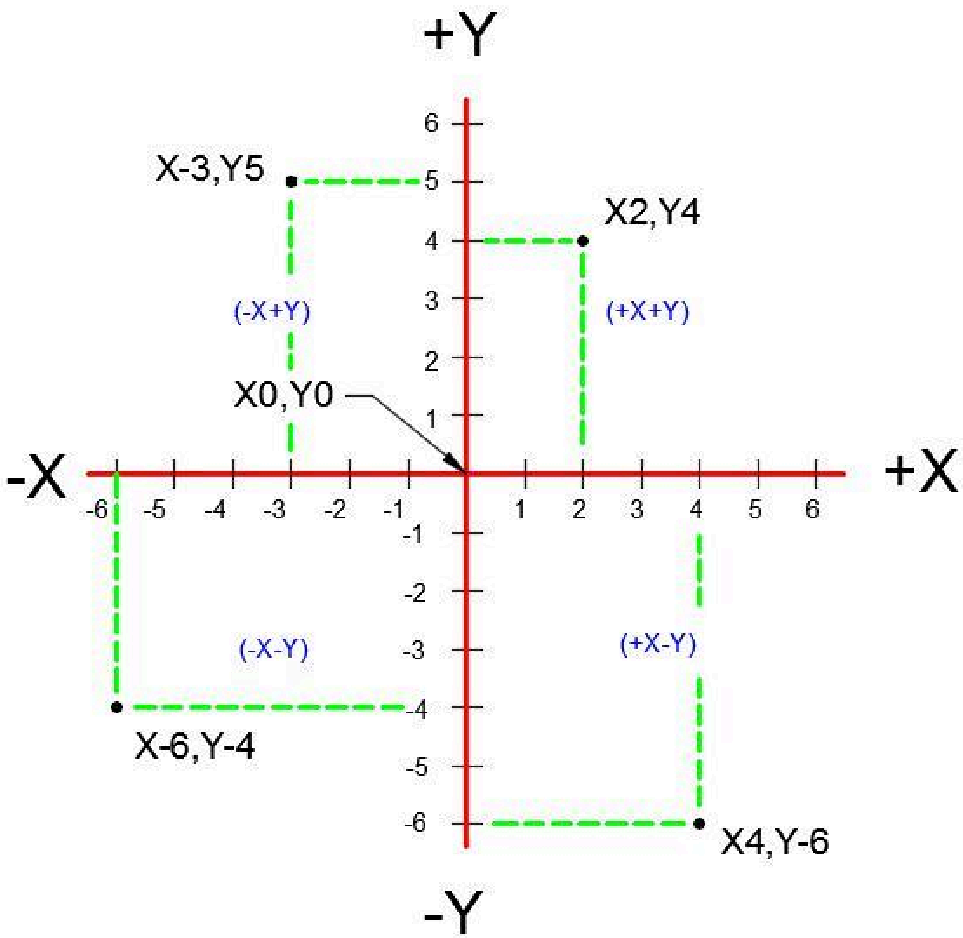


Figure 4-3
The Cartesian Coordinate System [Click to see image full size]

The *Cartesian Coordinate System* consists of two numbered lines crossing perpendicular to one another at their zero values. The horizontal axis is the X axis and the vertical axis is the Y axis. See Figure 4-3. A coordinate value is assigned to each location on the current construction plane. Each coordinate value consists of a pair of numbers, the first is the X coordinate and the second is the Y coordinate, written X,Y. The X and Y values are separated by a comma. For example, X2,Y4 is the location 2 units to the right and 4 units up from X0,Y0 or 0,0.

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

Parametric Solid Modeling

Inventor is a true Three Dimensional Parametric Solid Modeling system. A *parametric solid model* is a 3 dimensional solid model designed with geometrical and dimensional constraints rather then hard dimensions. A model designed in this way can then be modified by changing the dimensions, and/or constraints during or after the design is complete. When one or more constraints are modified, all other dependant constraints will automatically modify the model to conform to the new constraints. Geometric constraints are taught in this modules and dimensional constraints are taught in Module 5.

Geometrical Constraints

Geometrical constraints are used to apply geometrical relationships to the objects in the 2D sketch. They specify the geometrical relationship that the objects have to the sketching plane and to one another. Relationships like horizontal, vertical, parallel, or perpendicular are used. By applying geometrical constraints the number of dimensional constraints required to fully constrain the model is reduced. Applying the correct geometrical constraints prevents unwanted changes to a feature when geometry or dimensions are modified.

Since the geometrical constraint symbol displays and the constraints are automatically applied as the sketch is drawn, you have control on which constraints are applied to the objects as you are drawing the sketch. Constraints can be added while creating the sketch or by editing the sketch after the solid model is constructed. The geometrical constraints taught in this module are shown in Figure 4-4. Additional geometrical constraints will be taught throughout the Inventor book.

USER TIP: If you draw an incorrect line, you can click the Undo icon to remove it and insert it again. See Module 2, Undo and Redo Commands. It is sometimes easier to undo rather than deleting the line. This will only work if you click Undo icon immediately after you inserted the line and have not exited the LINE command.

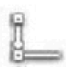
Geometrical Constraints			
Constraint	**Symbol**	**Icon**	**Definition**
Vertical	|		A vertical constraint is applied to a line that lies on or is parallel to the Y axis of the coordinate system.
Horizontal	—		A horizontal constraint is applied to a line that lies on or is parallel to the X axis of the coordinate system.
Perpendicular	⊥		A perpendicular constraint is applied to two lines that are at right angles to each other.
Parallel	//		A parallel constraint is applied to two lines that are parallel to each other.
Reference	None		A reference constraint is applied to reference geometry that is projected onto the sketching plane.
Coincidence	None		A coincidence constraint is applied to two points or endpoint of lines that are constrained to same point.

Figure 4-4
Geometrical Constraints

Projecting Reference Geometry

Projected reference geometry is geometry that has its position fixed relative to the sketching plane it resides on. Objects in the sketch are constrained with geometrical or dimensional constraints to the reference geometry to constrain it to the sketching plane. If the objects in the sketch are not constrained to the sketching plane, they will free float. A sketch that is not constrained to the sketching plane can never be fully constrained.

Inventor allows reference geometry to be projected in many different methods and uses. Many of these methods will be covered in the Inventor book. In this module, projecting the Center Point onto the plane of the Base sketch is taught.

Snapping

It is absolutely imperative that, when required, you snap to grids or locations on objects when drawing 2D sketches. Snapping to these locations ensures that the sketch is drawn accurately and constrained correctly. Inventor has many different snapping locations and they will be taught throughout the

Inventor book. For now, the snap locations shown in Figure 4-5 should be used when drawing sketches. Study the figure before starting the workalong.

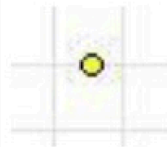
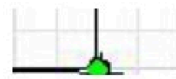
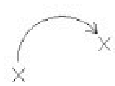
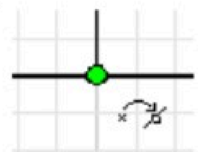
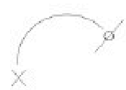
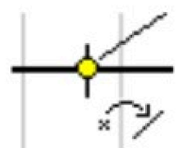
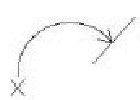
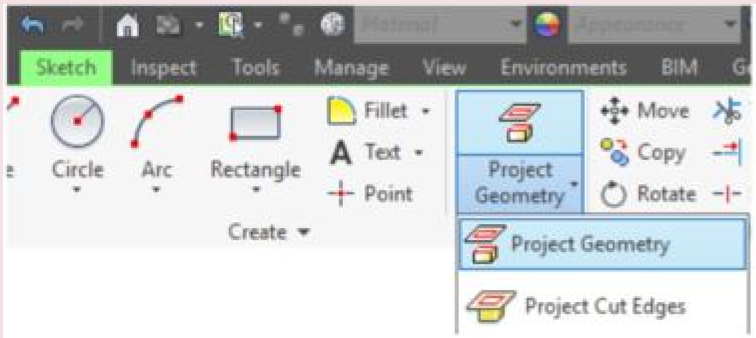
Snap Locations			
Mode	**Icon**	**How they appear in Inventor**	
Grid - Location in space (Snapping to a grid location)			Grid location but not on to an object.
Endpoint - Midpoint - Point (Snapping exactly onto the endpoint of an existing object)			Snapping to the endpoint of an object.
Endpoint Icon (Snapping onto the endpoint of an existing object)			Snapping to the midpoint of a line
Midpoint Icon (Snapping onto the midpoint of an existing object)			Snapping onto an object but not at any specific location on the object.
On the Object Icon (Snapping onto an existing object but not an exact location on the object)			

Figure 4-5
Snap Locations and Symbols

Inventor Command: PROJECT GEOMETRY

The PROJECT GEOMETRY command is used to project geometry to a fixed position on the 2D sketch plane.

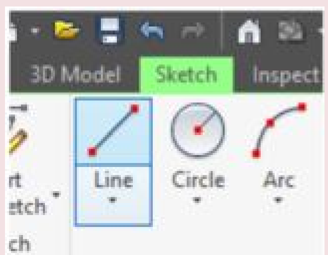
Shortcut: **none**



Inventor Command: LINE

The LINE command is used to draw lines on a sketch.

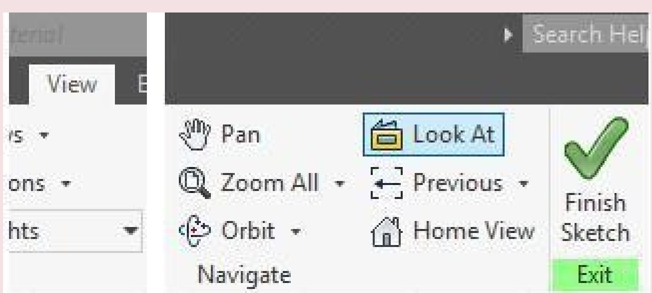
Shortcut: **L**



Inventor Command: LOOK AT

The LOOK AT command is used to change the users viewpoint to view the model or sketch perpendicular to the selected object, edge, or plane.

Shortcut: **PAGE UP**



Lines

Lines are the drawing objects that are used the most when drawing 2D sketches. A line is defined as the shortest distance between two XY coordinate locations. Once a line is drawn, Inventor knows the location of its endpoints as well as the midpoint of the line. Other lines or drawing objects can be drawn by snapping to those locations. See Figure 4-6.

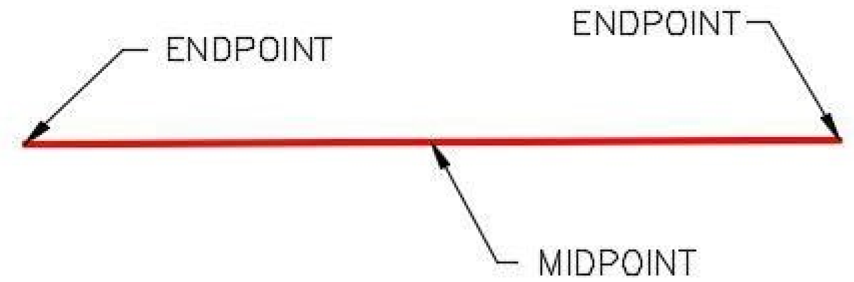


Figure 4-6
A Line

Base Sketch

The *Base sketch* is the first 2D sketch drawn in a new part. After the Base sketch is complete, it is extruded or revolved to create the Base model. Before drawing the Base sketch, you should study the model being constructed to determine the best plane to draw it on. The best view to use is the view with the most complex contour shape that does not contain arcs and curves. Draw the lines using lengths close to the finished dimensions. They do not have to be 100% accurate in length. The following rules should be followed when drawing the base sketch.

1. The objects in the sketch must meet exactly at their endpoints and cannot overlap.
2. The objects must form a perfect closed polygon and cannot contain any gaps.
3. The objects must have geometrical constraints applied to control the shape of the sketch.
4. Leave fillets and chamfers out of the original sketch. They can be added to the model after it is created. This is taught in Module 15.

X0Y0Z0

The two bold grid lines on the sketching plane represents the X and Y axis of the Cartesian Coordinate System. The axis lines display can be either enabled or disabled. It should currently be enabled. The horizontal line is the X axis and the vertical line is the Y axis. The point where they intersect is X0Y0Z0 of the sketch. Z is always zero since it is a 2D sketch.

The location of X0Y0Z0 on the Base sketch is a very important. You should pick its location on sketch carefully since it dictates the location of X0Y0Z0 on the completed model. See Figure 4-7.

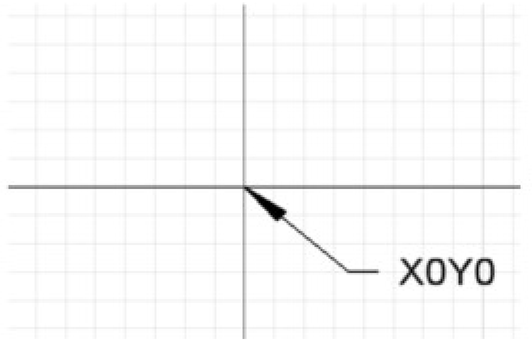


Figure 4-7
X0Y0 Origin Location

The Coordinate System Indicator shows the X, Y, and Z axis. The arrows always point in the positive X, Y, and Z direction. The red arrow is the X axis , the green arrow is the Y axis, the blue arrow is the z axis. See Figure 4-8 and 4-9.

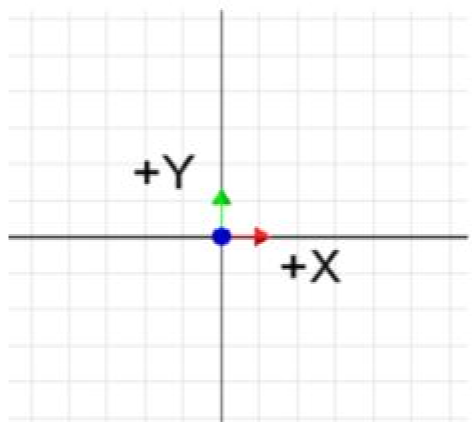


Figure 4-8
Coordinate System Indicator

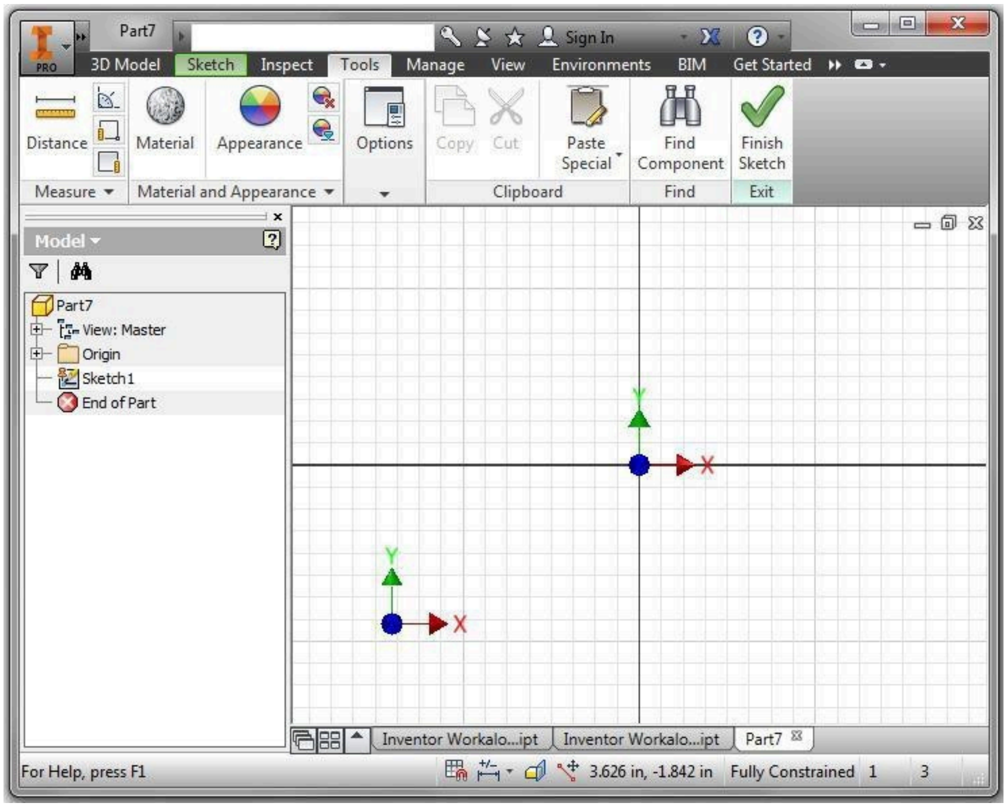
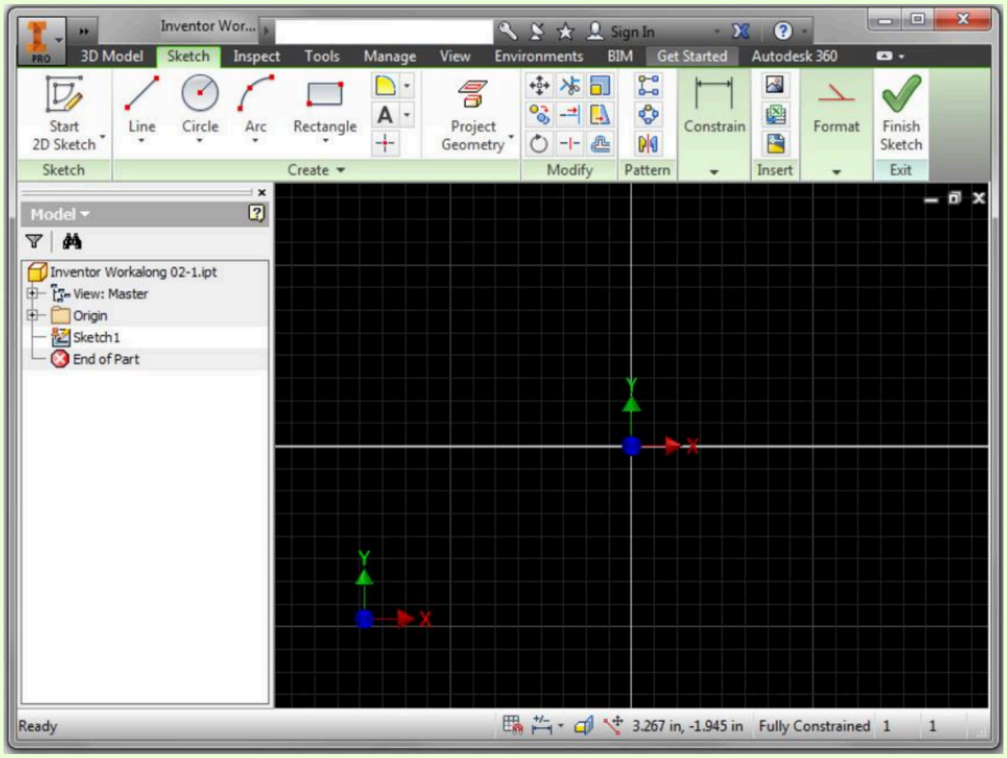
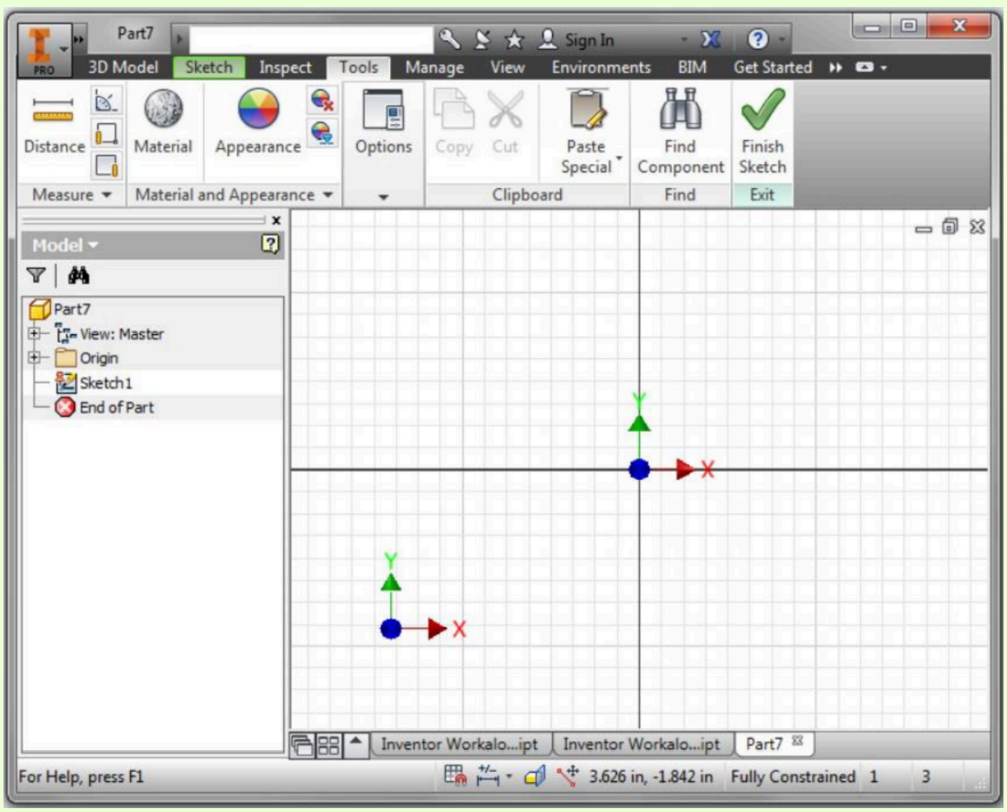


Figure 4-9
The Graphic window in 2D Sketch Mode [Click to see image full size]

AUTHOR'S COMMENTS: Your Graphic window should appear black as shown in the upper figure below while the Graphic window figures shown in future modules in the Inventor book will have a white background as shown in the lower figure. I used the white background in the book because they displays better.



[Click to see image full size]



[Click to see image full size]

MUST KNOW: Inventor allows files to be created in the units of inches, feet, millimeters, centimeters, meters, and microns. In the Inventor book, the units that are used are either inches or millimeters. When you start a new file, the template file you select sets the default units for that file.

WORK ALONG: Drawing the Base Sketch

Step 1

Start Inventor. Ensure that the current project is Inventor Course.

Step 2

Click the NEW command and start a new part using the template: English-Modules Part (in).ipt.

Step 3

Change to Model mode by clicking in the Finish sketch icon.

Step 4

Save the part file with the name: Inventor Workalong 04-1. The Graphic window should appear similar to the figure. (Figure Step 4)

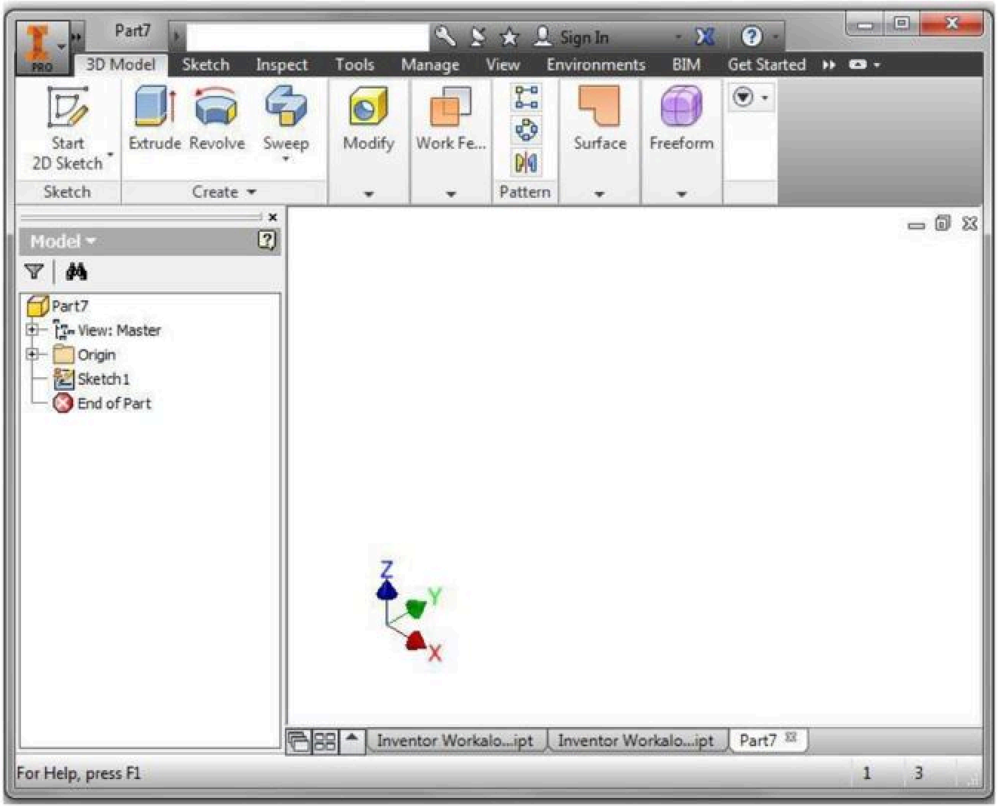


Figure Step 4 [Click to see image full size]

Step 5

Change to Sketch mode by right clicking Sketch1 in the Browser bar and click Edit Sketch. (Figure Step 5A and 5B)

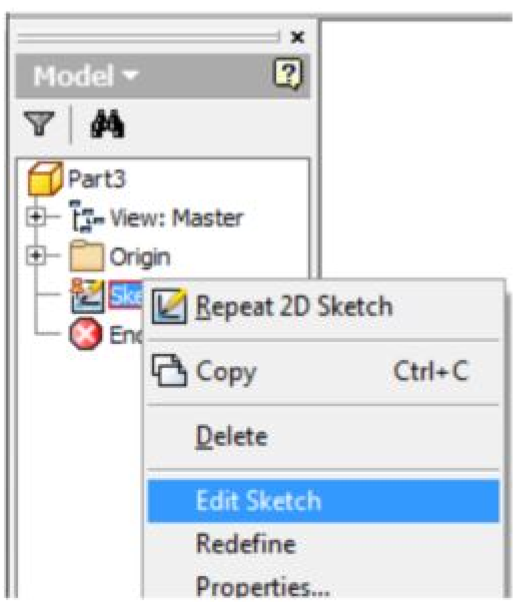


Figure Step 5A

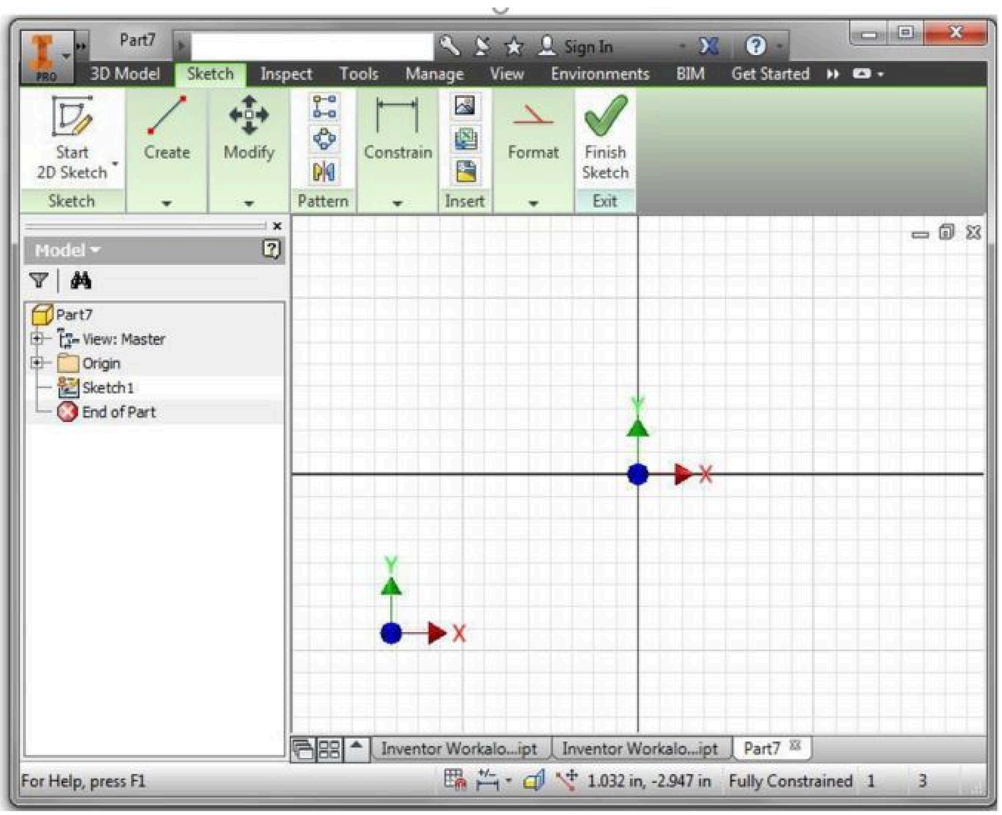


Figure Step 5B [Click to see image full size]

AUTHOR'S COMMENTS: The Graphic window is now in Sketch mode.

Step 6

The 3D model that you are constructing in this workalong is shown in the figure. (Figure Step 6A)

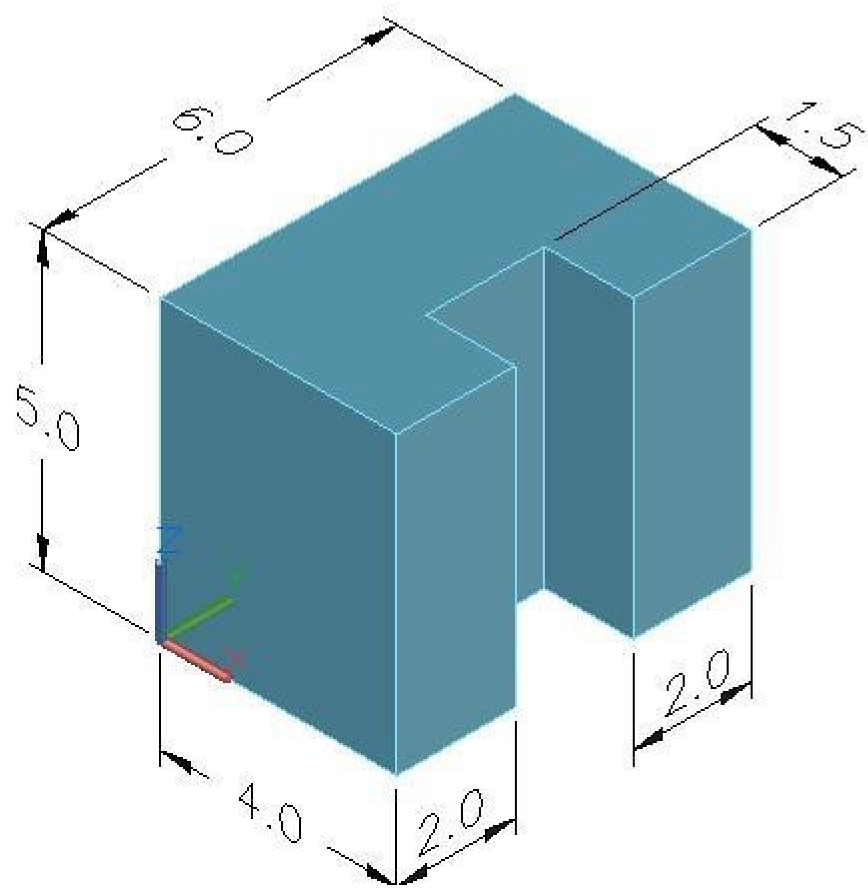


Figure Step 6A

AUTHOR'S COMMENTS: Note the location of X0Y0Z0 icon. The icon's vertex is located at X0Y0Z0. The red arrow points in the positive X direction, the green arrow points in the positive Y direction, and the blue arrow points in the positive Z direction.

AUTHOR'S COMMENTS: The first thing that you must do before you draw the Base sketch is to select the view with the most complex contour of the model. In this model, it is the top view as shown in Figure Step 6B.

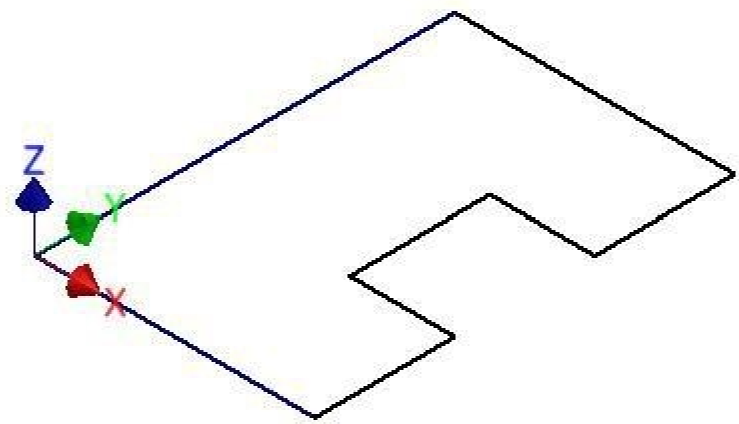


Figure Step 6B

Step 7

Press F6 to change to the Home view. (Figure Step 7)

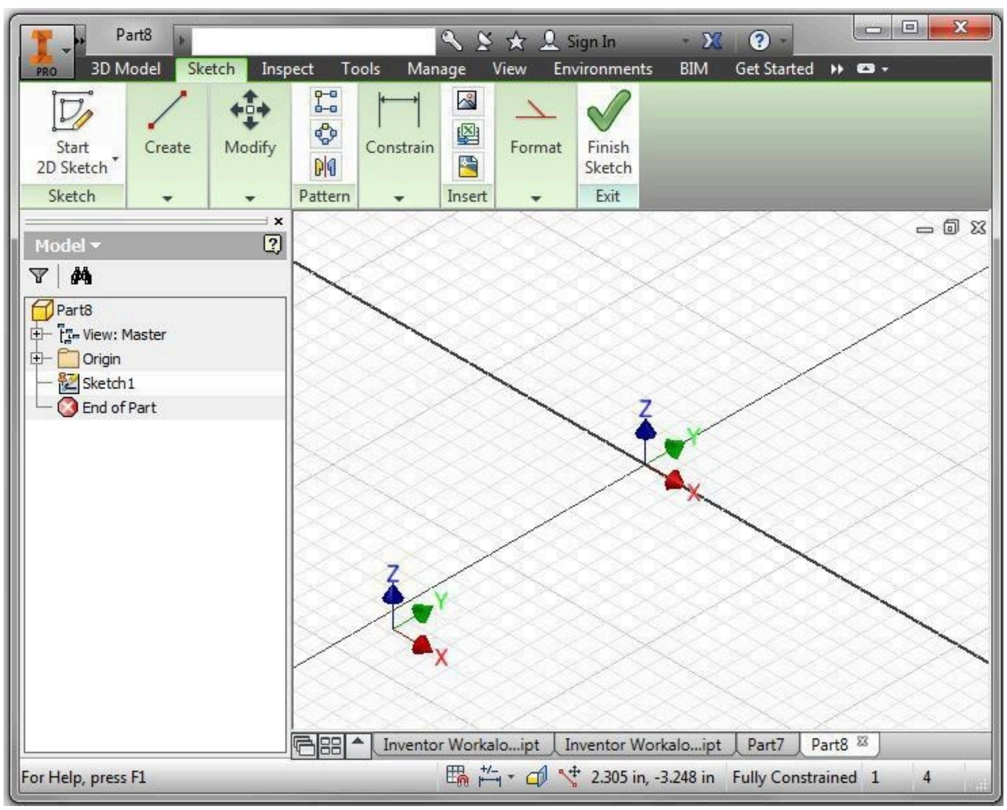


Figure Step 7 [Click to see image full size]

AUTHOR'S COMMENTS: You can draw a sketch in a 2D view or a 3D view. I show in the book drawing in different views. You can draw the sketch in the view that works best for you.

Step 8

Click the PROJECT GEOMETRY command. Note the Status bar, it displays the command prompt. Expand the Origin folder in the Browser bar and click Center Point. Press Esc to end the command and note that after you do that, the Status bar displays the Ready prompt. That means there is no current command. (Figure Step 8A, 8B, 8C, and 8D)

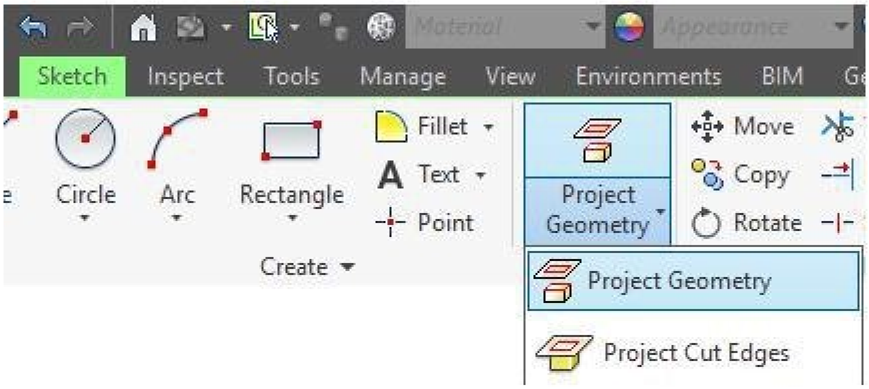


Figure Step 8A

Figure Step 8B

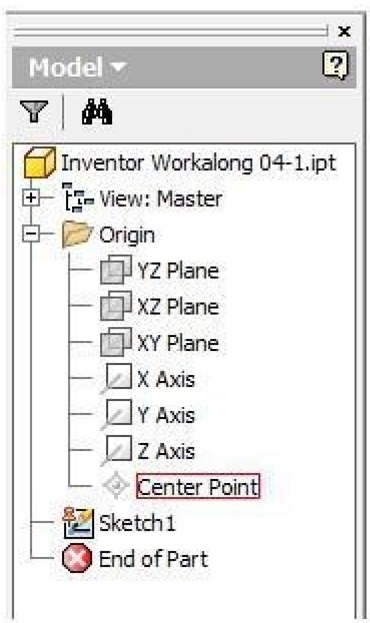


Figure Step 8C

Figure Step 8D

Step 9

Click the LOOK AT command. Move the cursor onto the Browser bar and click the XY Plane. The Graphic window will change to display the top view. Press Esc to end the command. (Figure Step 9A, 9B, and 9C)

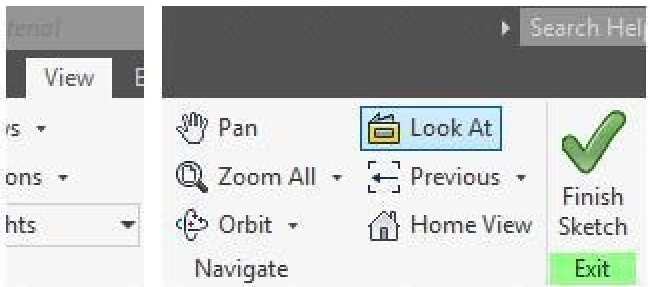


Figure Step 9A

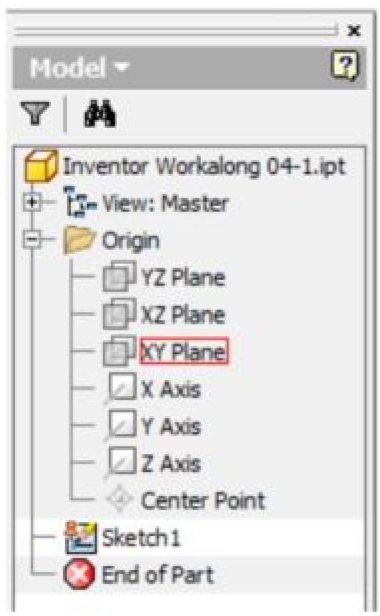


Figure Step 9B

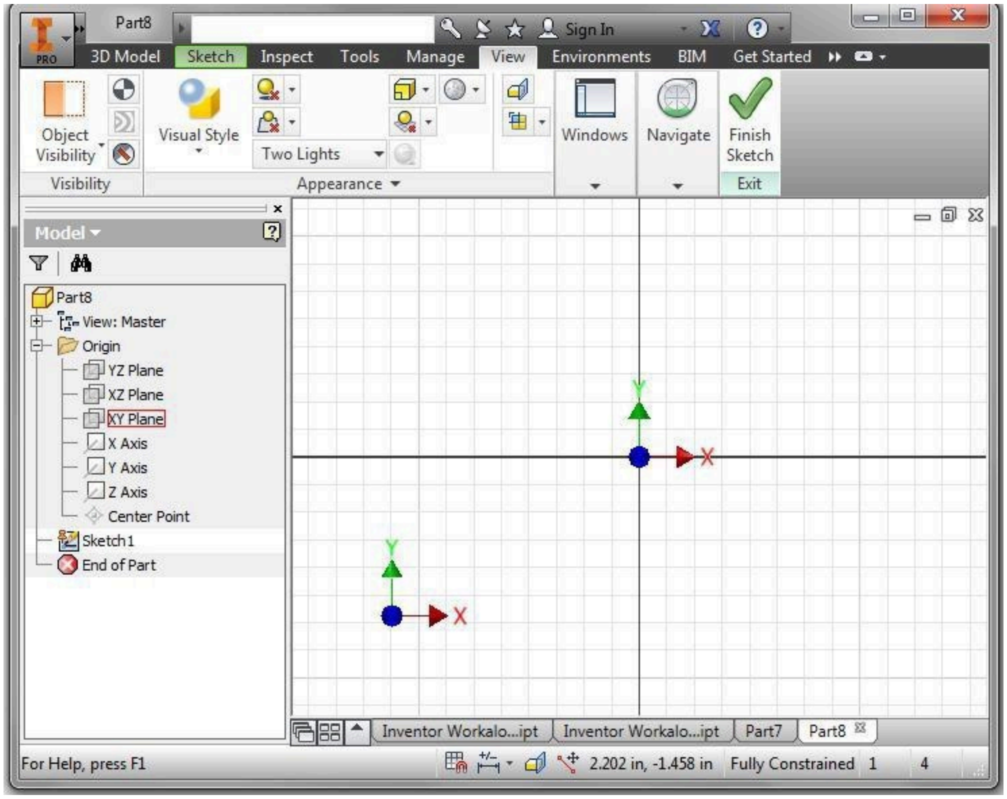


Figure Step 9C [Click to see image full size]

Step 10

Press F8. It enables the display of the geometrical constraint icons. To this point, there is only one geometrical constraint applied to sketch. It is the Reference constrain that was applied when you projected the Center Point using the PROJECT GEOMETRY command in Step 8. (Figure Step 10)

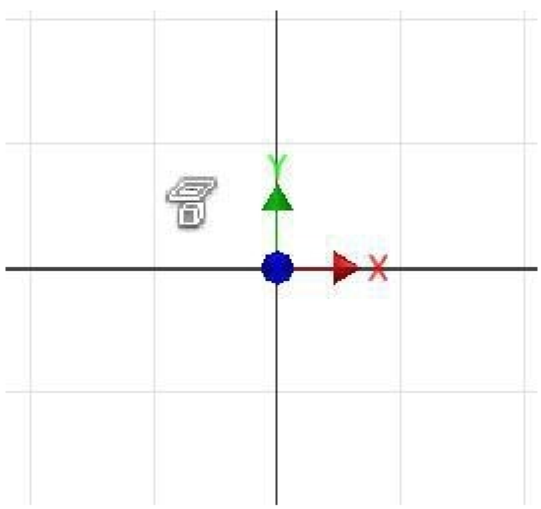


Figure Step 10

AUTHOR'S COMMENTS: The Reference constraint indicates that the Center Point is fixed to a position on the 2D sketch plane. That means that it cannot be moved on the plane and fully constrains the sketch up to this point.

Step 11

Press F9 to disable the display of the geometrical constraint icons.

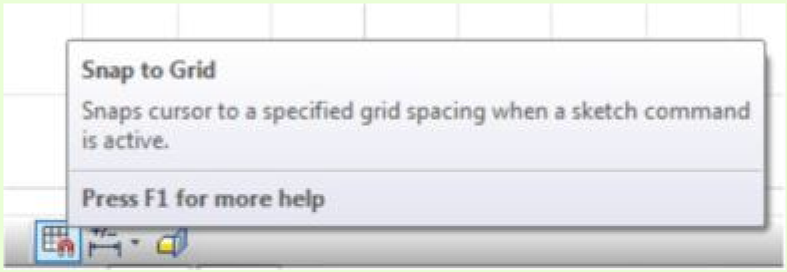
AUTHOR'S COMMENTS: In some computers, other background functions may have taken over the F keys. In that case, right- click somewhere in the Graphics window and then click on Hide All Constraints.

Step 12

Move the cursor somewhere in the Graphic window and right click the mouse. In the Right-click menu, ensure that Snap to Grid is enabled. (Figure Step 12)

Figure Step 12

AUTHOR'S COMMENTS: Snap to Grid can be also enabled in the Status bar as shown below.



Step 13

Click the LINE command. Note the Status bar prompt. Move the cursor to X0Y0Z0 and hold it there. A green snap point will display. That means it is snapping to the Center Point geometry that was projected in Step 8. While the green snap point is displayed, click the left mouse button. (Figure Step 13A, 13B, and 13C)

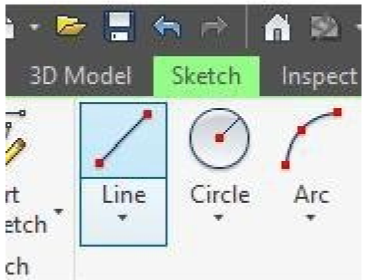


Figure Step 13A

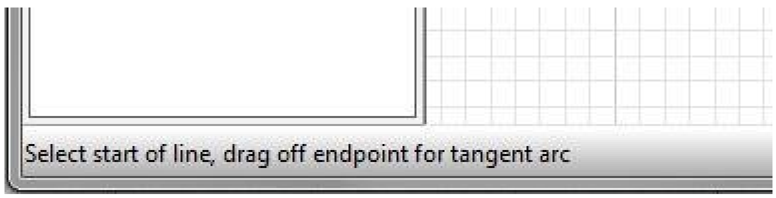


Figure Step 13B

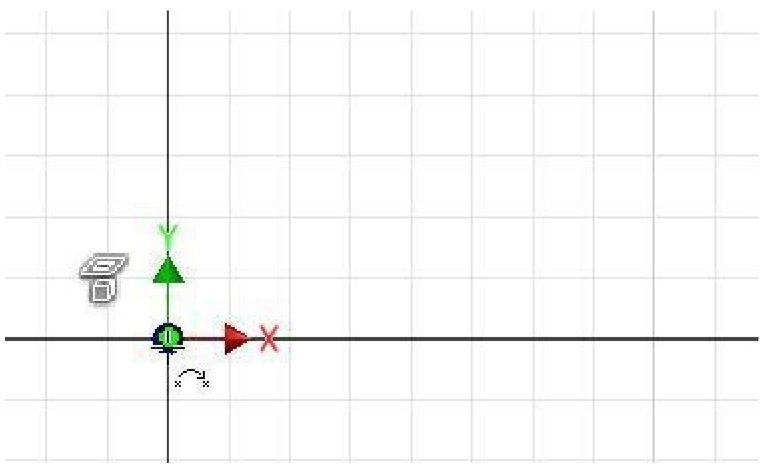


Figure Step 13C

AUTHOR'S COMMENTS: When snapping to a point in Inventor, it is important to move slowly. If you move the mouse too quickly after clicking the left mouse button, you may miss snapping exactly to the desired point.

Step 14

Move the cursor about 4 inches along the positive X axis. Watch the Status bar and it will display the length of the line as the cursor is moved. When it is about 4 inches long and the Horizontal geometrical constraint symbol and the yellow snap point displays, click the left mouse button. (Figure Step 14A and 14B)

Figure Step 14A

Figure Step 14B

AUTHOR'S COMMENTS: When snapping to a point in Inventor, it is important to move slowly. If you move the mouse too quickly after clicking the left mouse button, you may miss snapping exactly to the desired point.

Step 15

Move the cursor about 2 inches in the positive Y direction. When the Perpendicular geometrical constraint symbol displays at the same time as the yellow grid snap point, click the left mouse button. (Figure Step 15A and 15B)

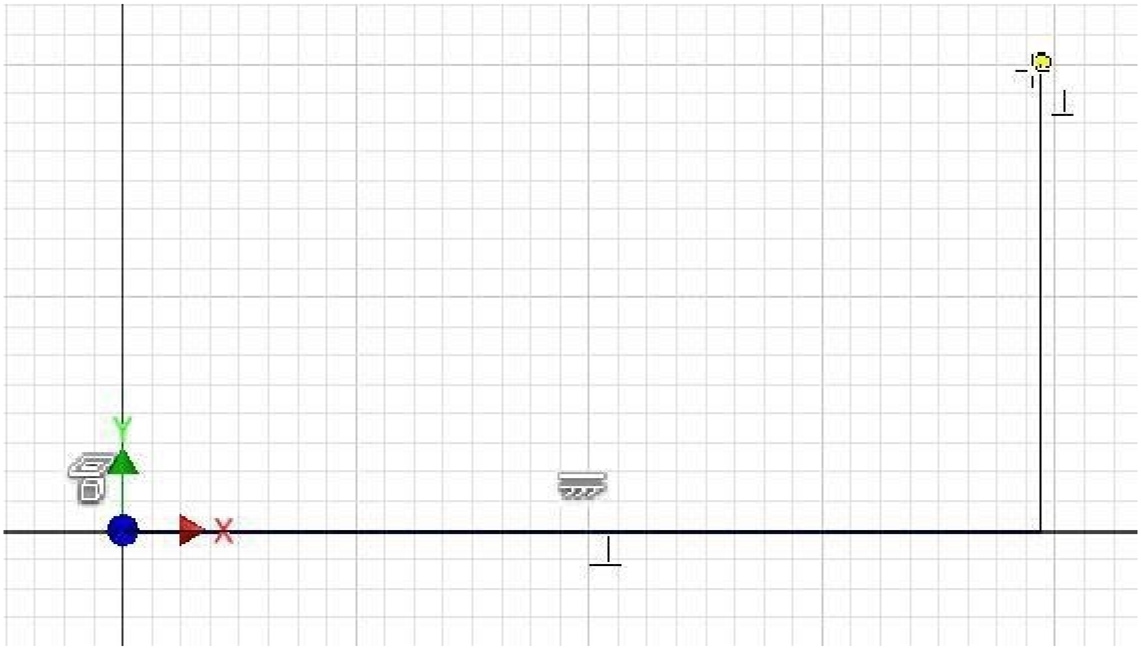


Figure Step 15A

Figure Step 15B

AUTHOR'S COMMENTS: Move slow and deliberate when snapping to objects or grids.

Step 16

Move the cursor about 2 inches in the negative X direction. When the Perpendicular constraint symbol displays and the yellow snap grid point displays, click the left mouse button. (Figure Step 16A and 16B)

Figure Step 16A

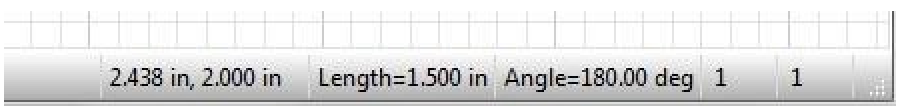


Figure Step 16B

AUTHOR'S COMMENTS: You do not have to draw lines to their exact length. Just keep them close. It is more important that you are snapping to points and applying the applicable geometrical constraints.

Step 17

Move the cursor about 2 inches in the positive Y direction. When the Parallel constraint symbol and the yellow snap grid point displays, click the left mouse button. (Figure Step 17A and 17B)

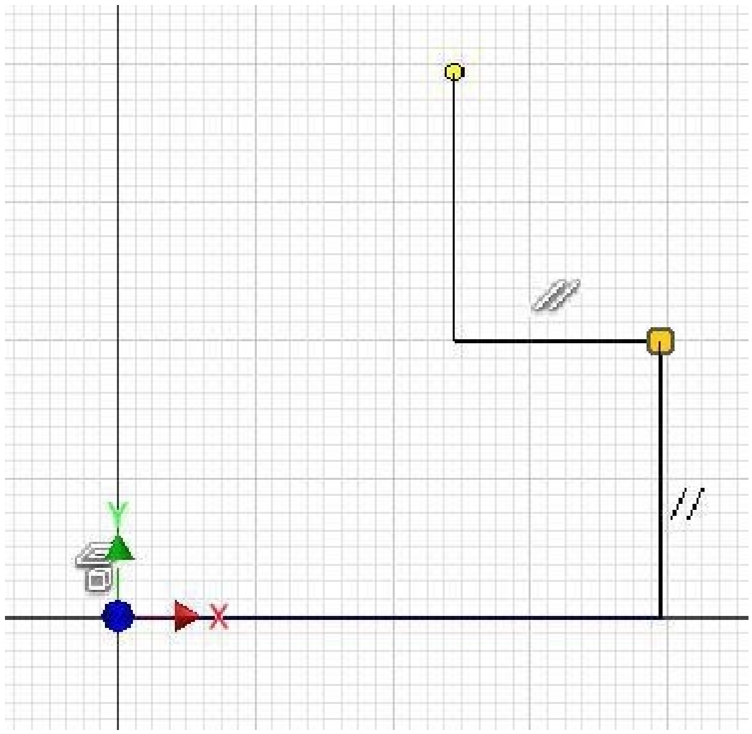


Figure Step 17A

Figure Step 17B

AUTHOR'S COMMENTS: Do not be concerned if the Perpendicular constraint is applied in your sketch. Either the Parallel or Perpendicular constrain will work in this sketch.

Step 18

Move the cursor about 2 inches in the positive X direction. When the Perpendicular constraint symbol and the yellow snap grid point displays, click the left mouse button. (Figure Step 18A and 18B)

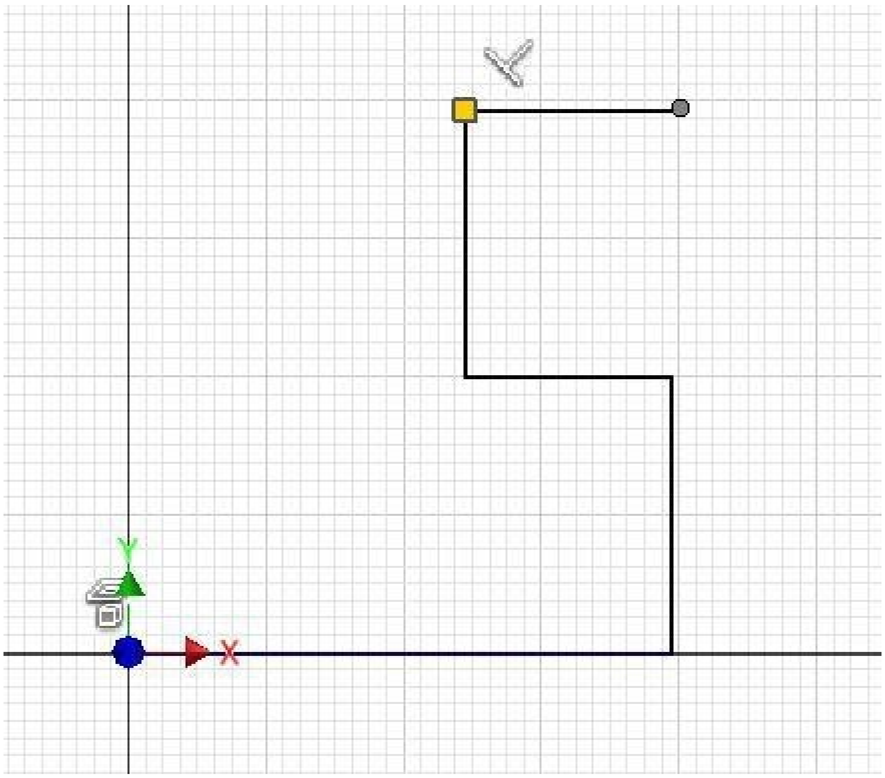


Figure Step 18A

Figure Step 18B

Step 19

Move the cursor about 2 inches in the positive Y direction. When the Perpendicular constraint symbol and the yellow snap grid point displays, click the left mouse button. (Figure Step 19)

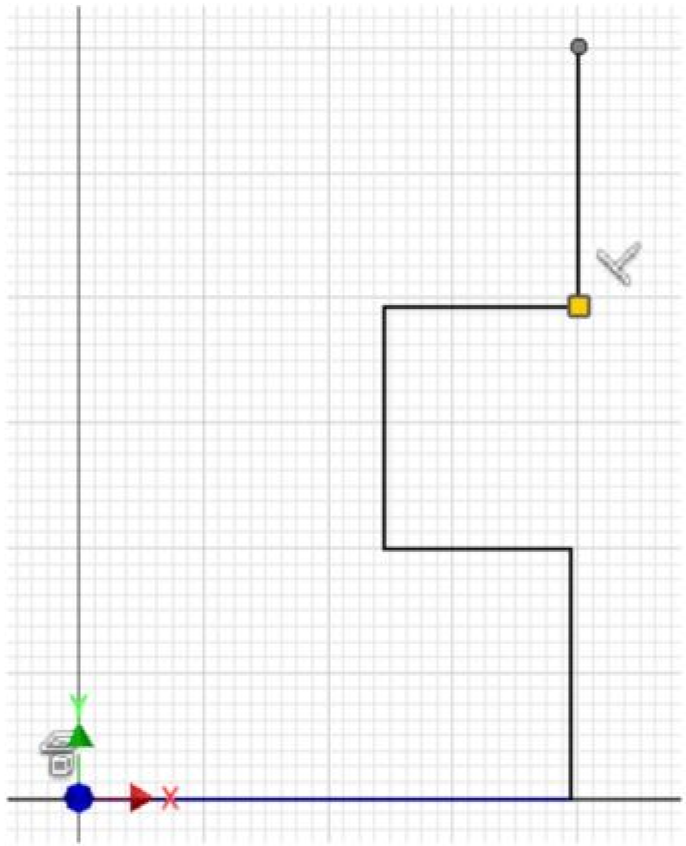


Figure Step 19

Step 20

Move the cursor about 4 inches in the negative X direction. When the Perpendicular constraint symbol and the yellow snap grid point displays, on the Y axis, click the left mouse button. (Figure Step 20)

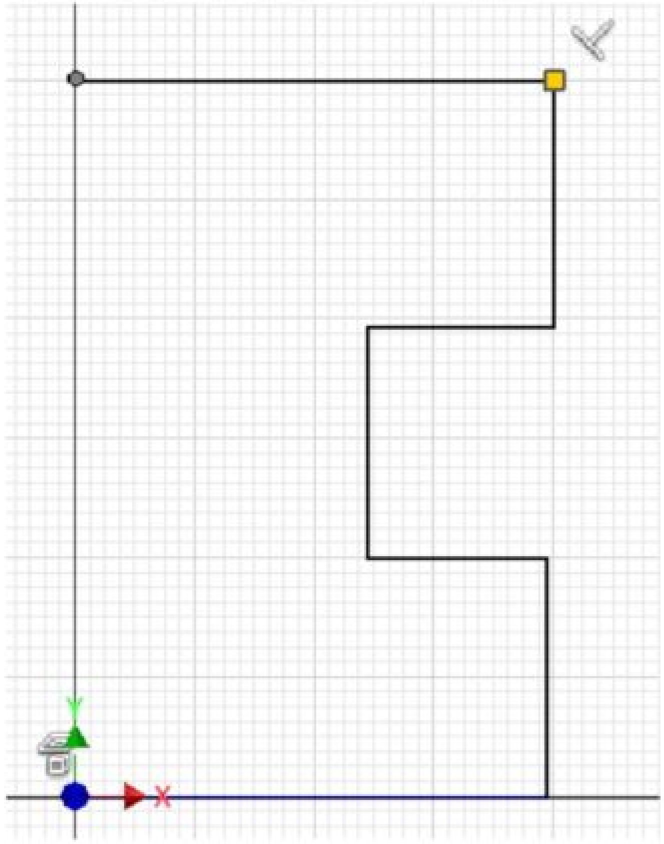


Figure Step 20

AUTHOR'S COMMENTS: Ensure that the yellow grid snap point displays on the Y axis.

Step 21

Move the cursor to X0Y0Z0. When the green snap point and the Perpendicular constraint symbol displays, click the left mouse button. (Figure Step 21)

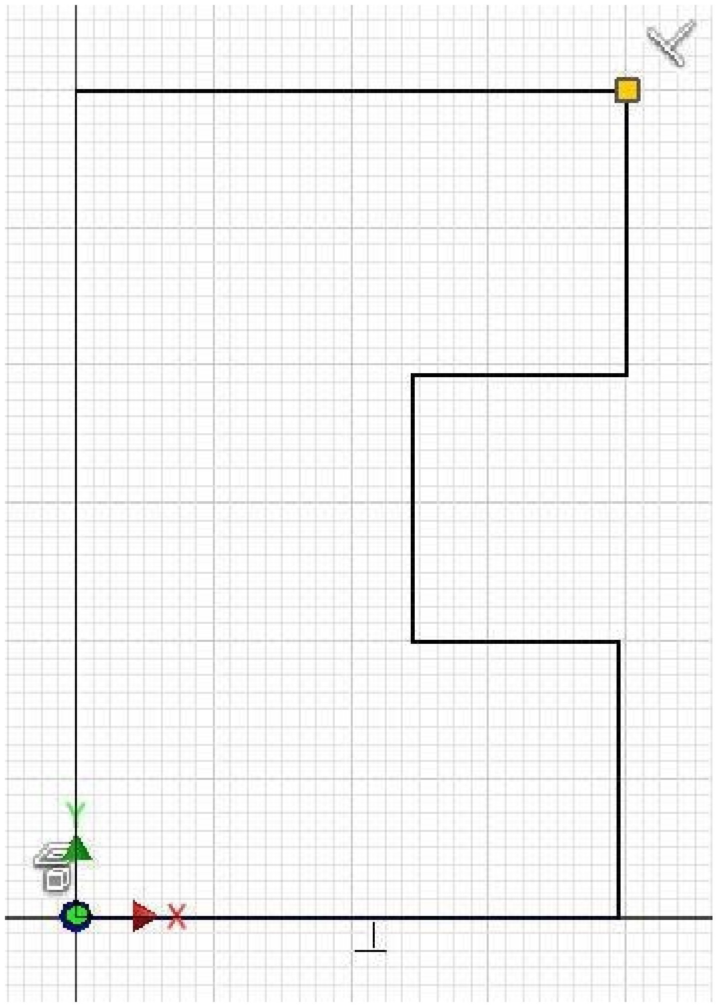


Figure Step 21

AUTHOR'S COMMENTS: Ensure that green snap point displays before pressing the left mouse button. The green snap point indicates that you are snapping to the projected Center Point.

Step 22

Press Esc to end the LINE command. Note that the Status bar prompts Ready meaning there is no current command. (Figure Step 22)

Figure Step 22

MUST KNOW: When the Snap to Grid feature is enabled, the points selected will snap to locations on the grid. As a beginner, it is best to draw by snapping to the grid to control the location and size of the sketch.

Step 23

Click the LOOK AT command and change the view to the top or XY plane. If you have trouble, see Step 9.

Step 24

Press F8 to display the geometrical constraint icons. They should appear similar to the figure. (Figure Step 24)

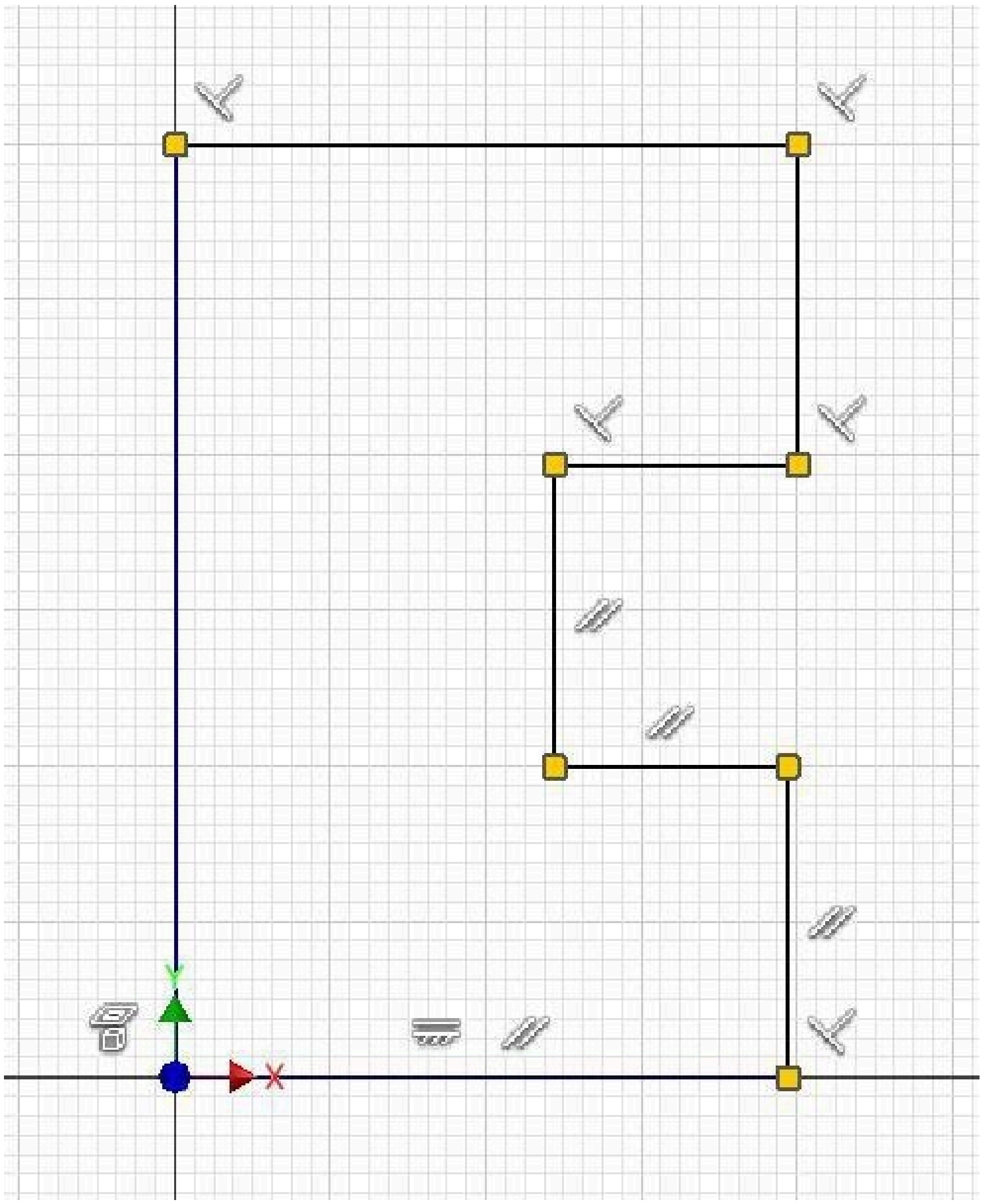


Figure Step 24

AUTHOR'S COMMENTS: The constraint icons in your sketch may not match the figure exactly.

MUST KNOW: A correctly drawn Base sketch must form a closed polygon. There can be no gaps. Each object in the sketch must meet exactly at their endpoints and cannot overlap.

Step 25

Move the cursor onto the Horizontal geometrical constraint icon. Note how the bottom line display red to indicate that the constraint is applied to that line. (Figure Step 25)

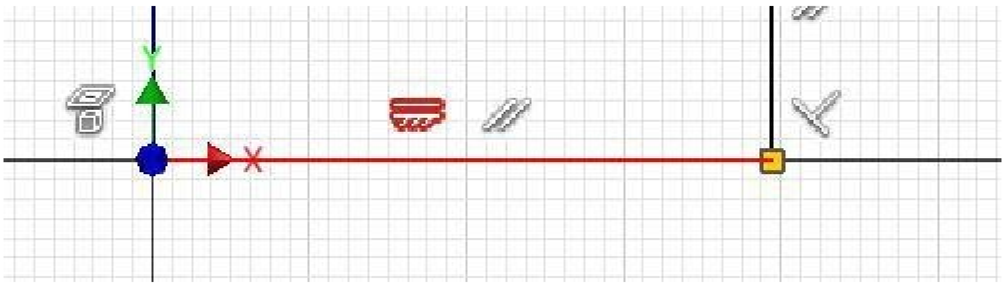


Figure Step 25

Step 26

Move the cursor onto a Parallel geometrical constraint icon and note how another Parallel geometrical constraint icon will highlight showing you which two icons match one another. Note how the two parallel lines will also highlight to indicate that the upper line is constrained to the lower line with a Parallel geometric constraint. (Figure Step 26)

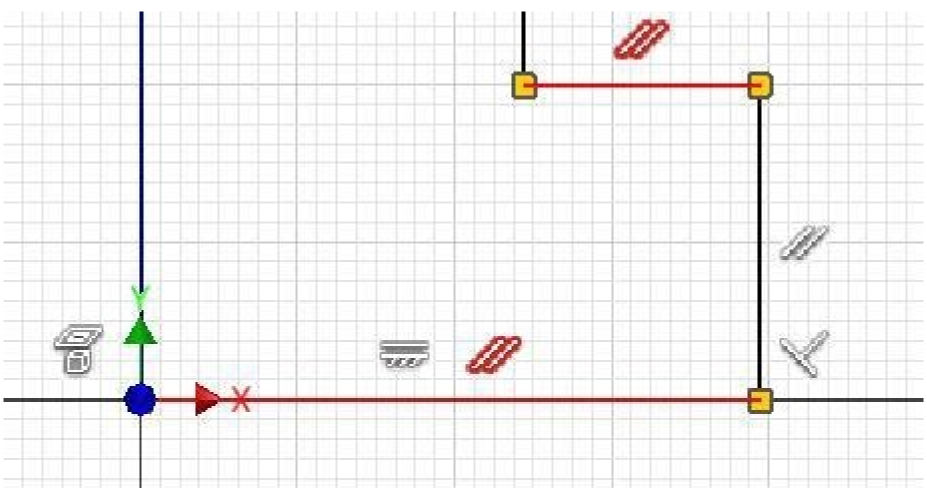


Figure Step 26

Step 27

Move the cursor onto the Perpendicular geometrical constraint icon. Note how the two lines highlight. The vertical line is constrained perpendicular to the horizontal line. (Figure Step 27)

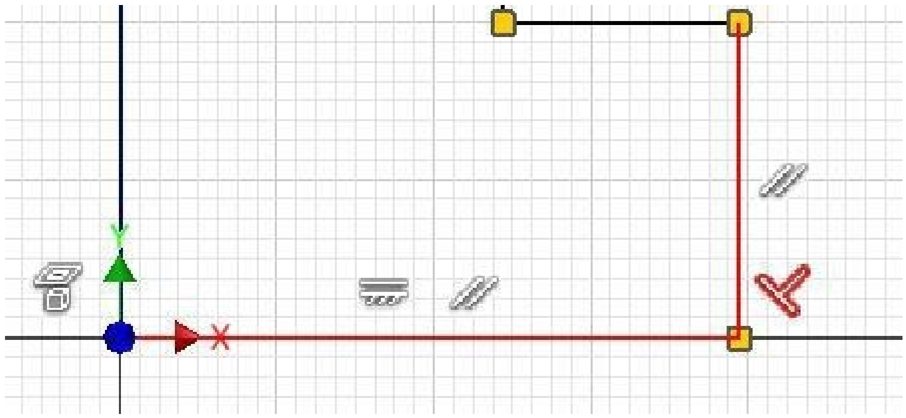


Figure Step 27

Step 28

Move the cursor onto the small square icon at the bottom right corner of the object. Note how two Coincident geometrical constraint icons will display indicating both line's endpoints are at the exact same XY location. (Figure Step 28)

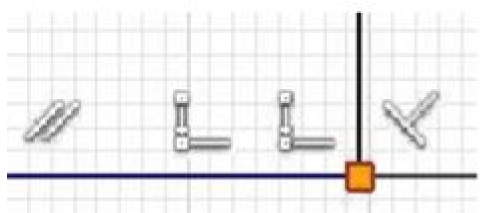


Figure Step 28

Step 29

Press F9 to disable the display of the geometrical constraint icons. Press F6 to display the Home view. The completed sketch should appear as shown in the figure. (Figure Step 29)

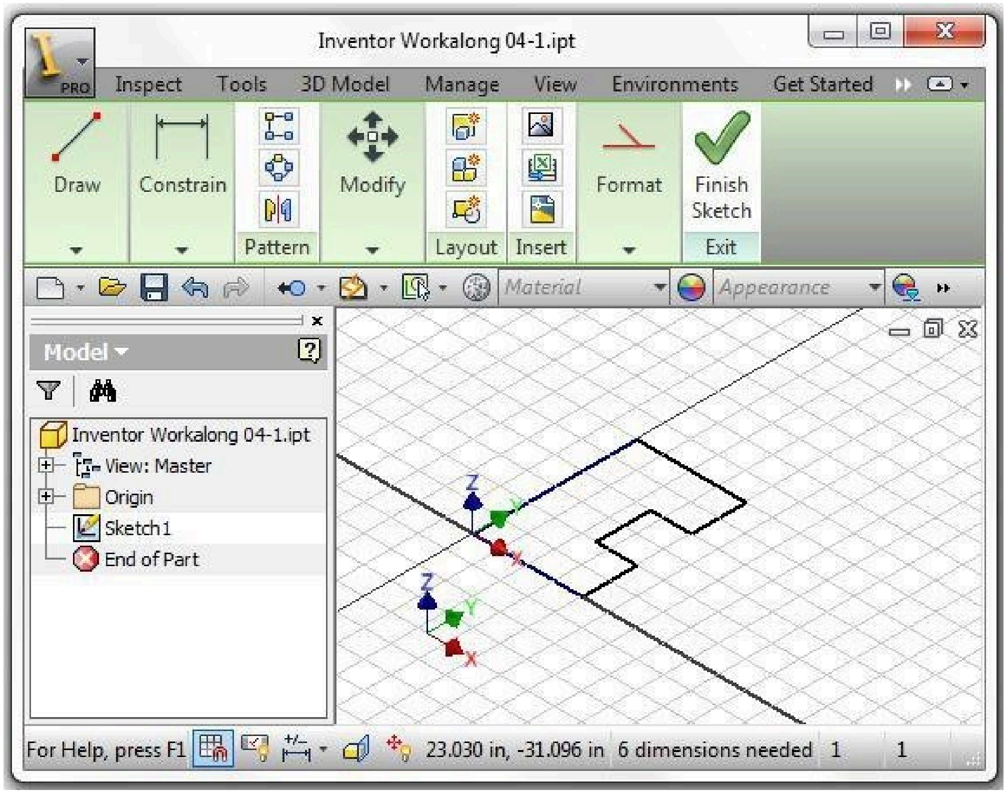


Figure Step 29 [Click to see image full size]

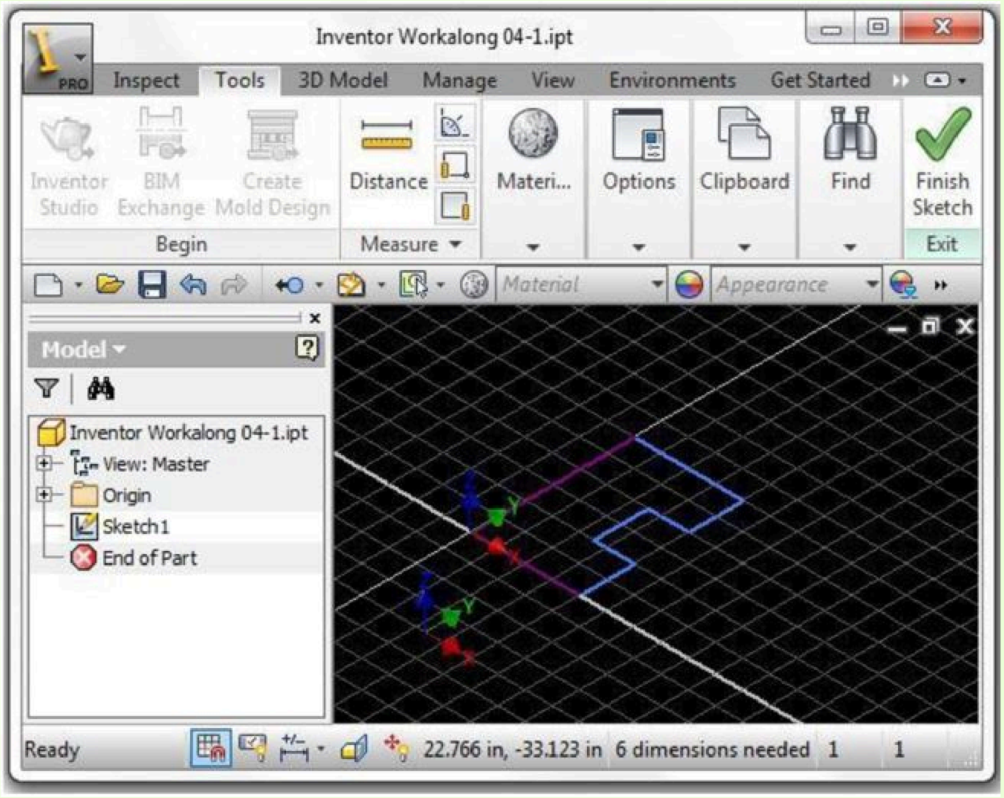
Step 30

Return to Model mode.

Step 31

Save and close the file.

AUTHOR'S COMMENTS: The final sketch is shown in black as your screen should. Note that two lines are purple and the remainder are blue. If the two lines do not display purple, redo the workalong. You will learn the importance of this in Module 5.



[Click to see image full size]

Deleting Lines

There are two methods to delete any unwanted drawing object.

Method 1

Ensure that there is no active command and without entering a command, select the drawing object with the cursor. If it is successfully selected, it will change colour. Right click the mouse. In the Right-click menu, click Delete as shown in Figure 4-9.

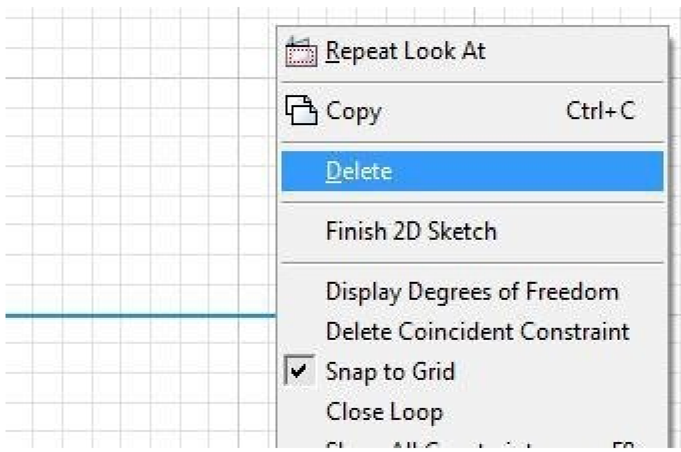


Figure 4-9
Deleting Lines

Method 2

Select the drawing object the same way as in Method 1. When the selected object changes colour, press the Delete key on the keyboard.

AUTHOR'S COMMENTS: Draw some random lines in Sketch mode. Practice deleting the lines a few times until you can do it easily.

Key Principles

Key Principles in Module 4

1. Inventor is a true Three Dimensional Parametric Solid Modeling system. A parametric solid model is a 3 dimensional solid model designed with parameters and constraints rather then hard dimensions.
2. Geometrical constraints are used to apply geometrical relationships on or between the objects in the 2D sketch. They specify the geometrical relationship the objects have to the sketching plane or one another.
3. It is absolutely imperative that, when required, you snap to grids or pre-defined locations on objects when drawing 2D sketches. Snapping ensures that the sketch is drawn accurately and constrained correctly.
4. The Base sketch is the first sketch drawn. The Base sketch is extruded or revolved to create the Base model. Before drawing the Base sketch, you should select the view with the most complex contour shape to draw the sketch on.
5. Projected geometry is geometry that has its position fixed relative to the sketching plane where it originates. Drawing objects in the sketch are constrained or dimensioned to the projected geometry to constrain it to the sketching plane.

Lab Exercise 4-1

Time allowed: 40 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab 04-1	Inventor Course	Inches	English-Modules Part (in).ipt	N/A	N/A

Step 1

Start a new part file with the template: English – Modules Part(in).ipt save the file with the name: Inventor Lab 04-1 as shown above.

Step 2

Project the Center Point onto the Base sketch.

Step 3

Draw the Base sketch for the 3D model shown below and apply all of the necessary geometrical constraints. Note the location of X0Y0Z0. (Figure Step 3A and 3B)

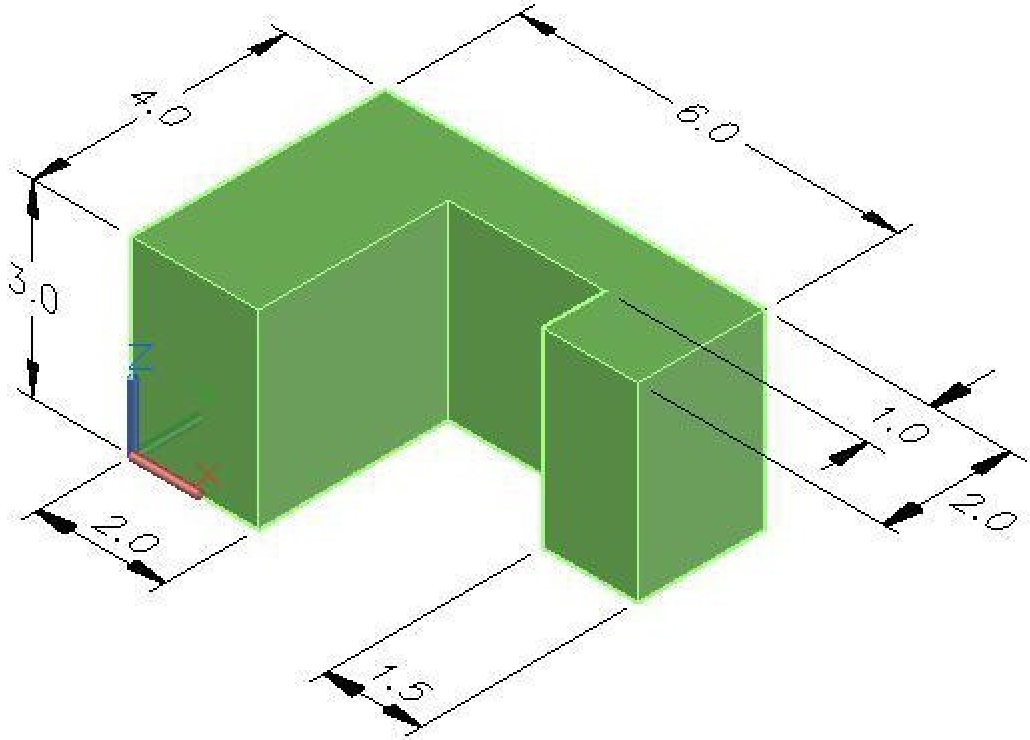


Figure Step 3A

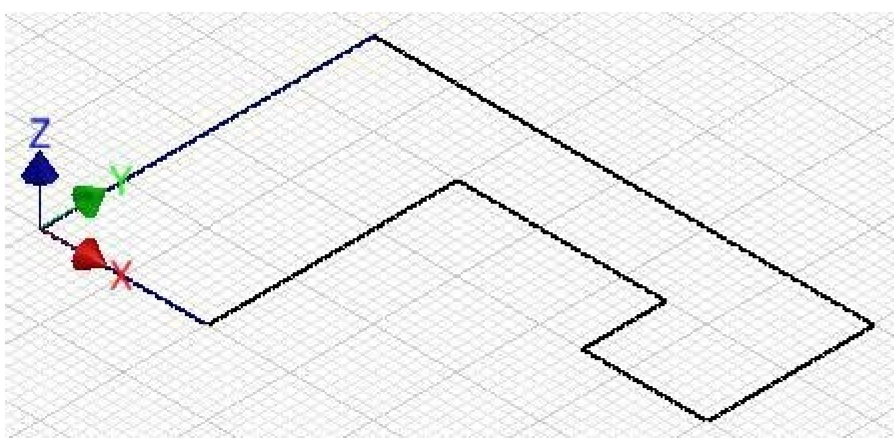
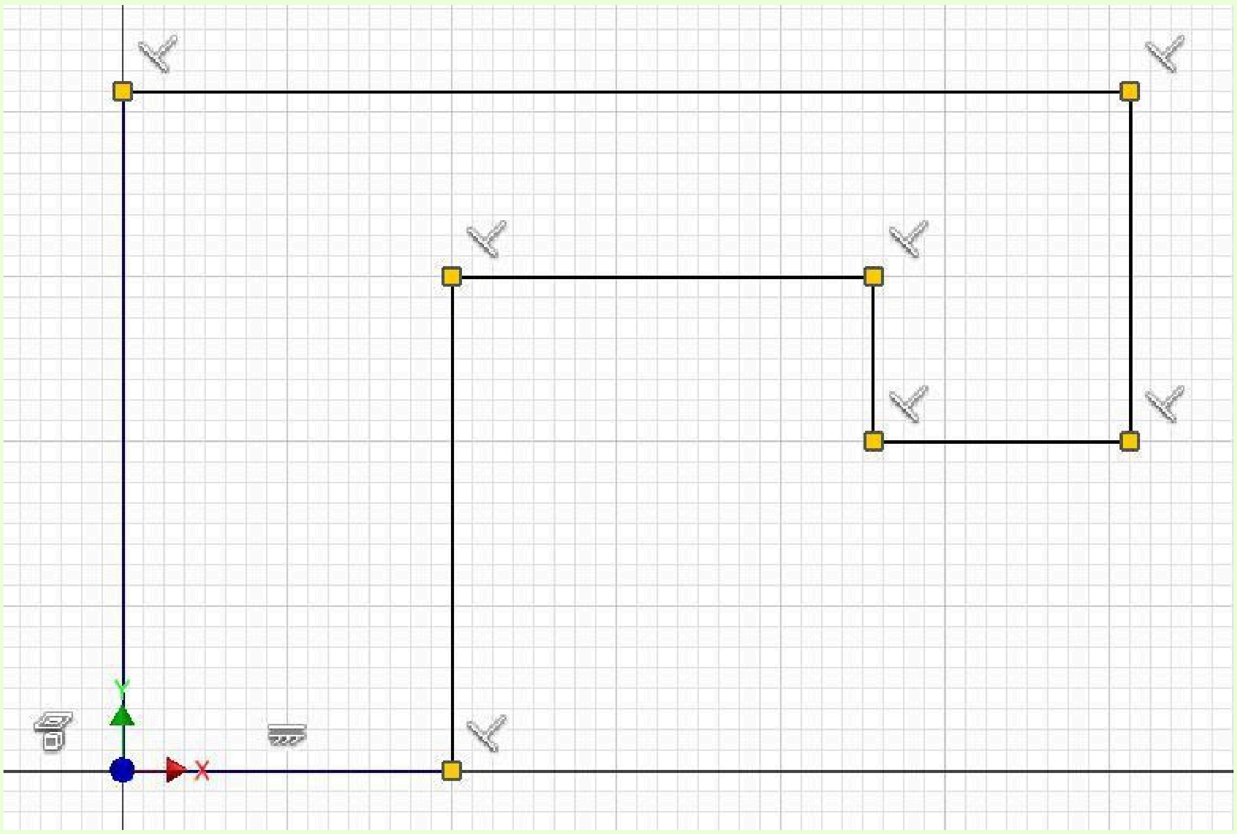


Figure Step 3B
Base Sketch

AUTHOR'S GEOMETRIC CONSTRAINS: The following figure shows the Base sketch's geometric constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketch using different construction methods and constraints, that is what is important. You may want to compare your construction method and constraints used with the authors.



Author's Base Sketch

Lab Exercise 4-2

Time allowed: 40 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab 04-2	Inventor Course	Inches	English-Modules Part (in).ipt	N/A	N/A

Step 1

Start a new part file with the template: English – Modules Part(in).ipt and save the file with the name: Inventor Lab 04-2 as shown above.

Step 2

Project the Center Point onto the Base sketch.

Step 3

Draw the Base sketch for the 3D model shown below and apply all of the necessary geometrical constraints. Note the location of X0Y0Z0. (Figure Step 3A and 3B)

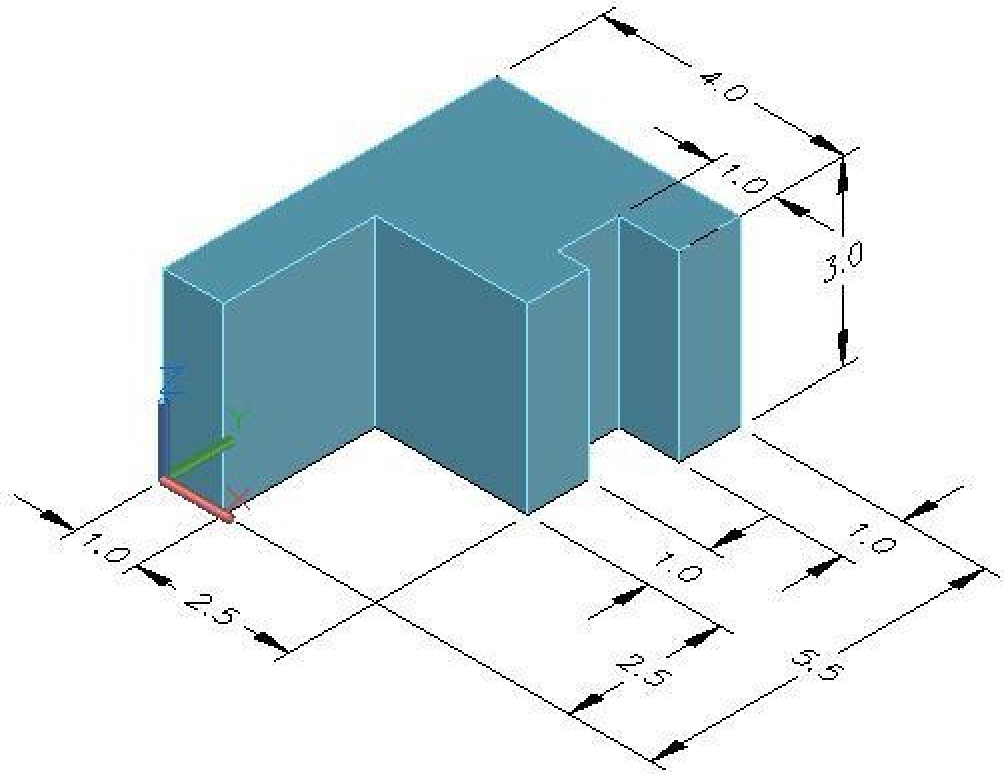


Figure Step 3A

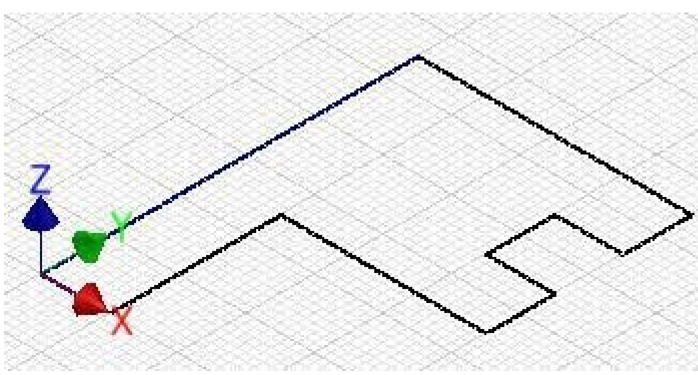
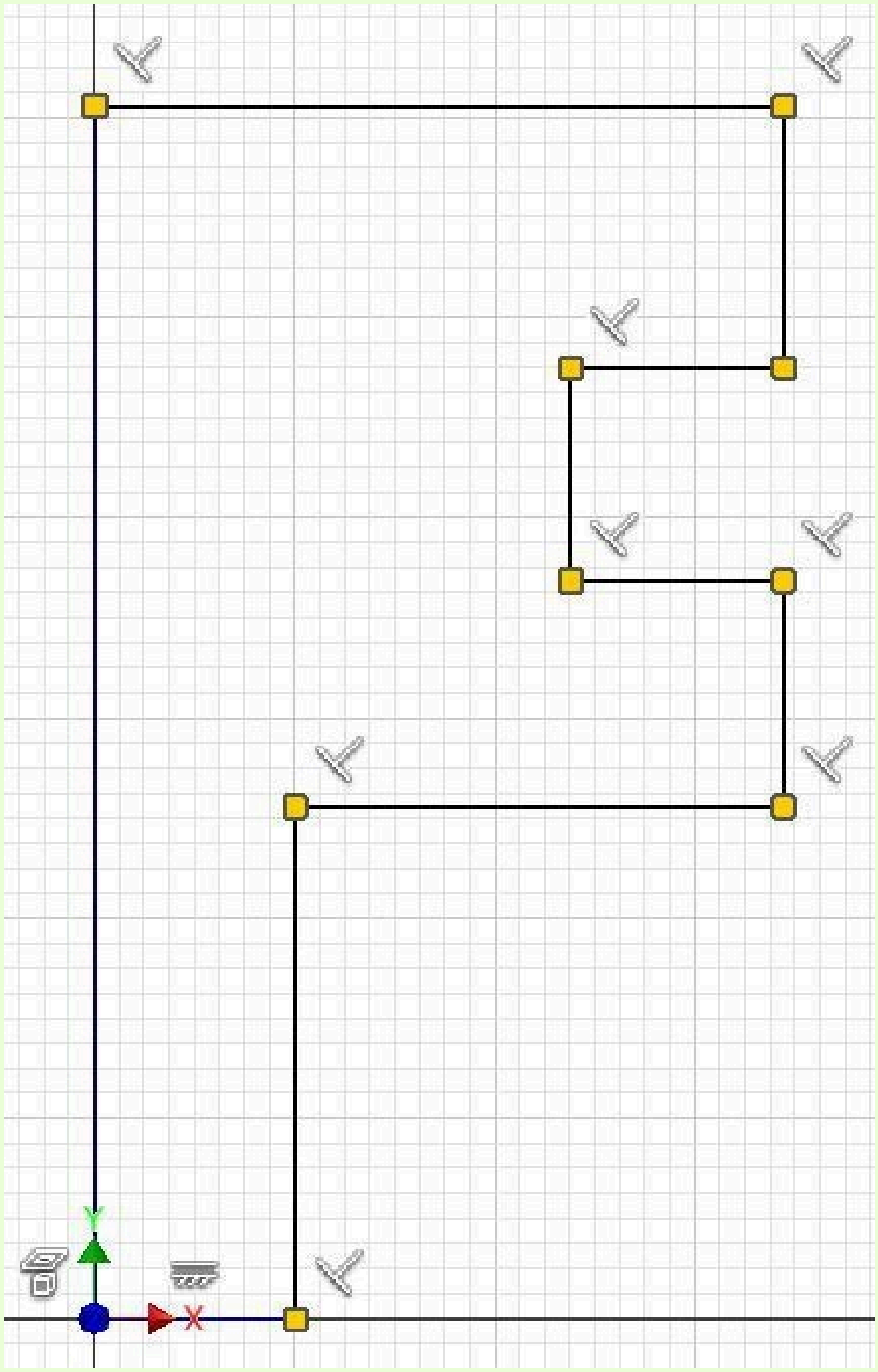


Figure Step 3B Base Sketch

AUTHOR'S GEOMETRIC CONSTRAINS: The following figure shows the Base sketch's geometric constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketch using different construction methods and constraints, that is what is important. You may want to compare your construction method and constraints used with the authors.



Author's Base Sketch

Module 5 Extruding – Part 1

Learning Outcomes

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

1. Describe dimensional constraints, linear dimensions, driving and driven dimensions.
2. Apply the GENERAL DIMENSION command to insert the necessary linear dimensions to fully constrain Base sketches.
3. Describe and apply the EXTRUDE command to extrude Base sketches to create the 3D Base model.

Dimensional Constraints

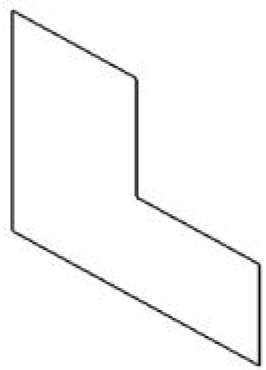
Unlike geometrical constraints, that are used to apply geometrical relationships between the objects in the sketch, *dimensional constraints* control and report the size of the geometry. Dimensional constraints are sometimes called *parametric dimensions*. To fully constrain a 2D sketch, driving dimensions must be applied. A *driving dimension* is a parametric dimension controlling the size of the object. Inventor will automatically change the overall model to conform to the driving dimensions maintaining the existing geometrical constraints that were assigned in the sketch. Objects of the sketch that are not dimensioned will change to adapt when a driving dimension is applied or an existing driving dimension is changed.

Add only the number of driving dimensions that are required to ensure that the model maintains the desired size and shape. Inventor will issue a warning when a dimension is added that over-constrains the sketch. Only driven dimensions will be allowed to be added to a fully constrained sketch.

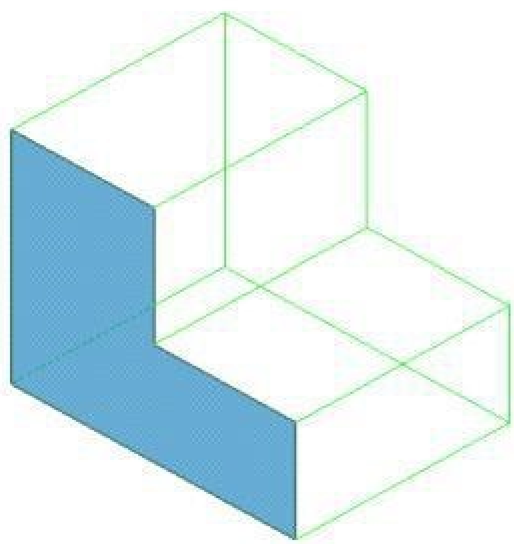
A *driven dimension* is a non-parametric dimension that does not constrain the object. It only displays the current value of the geometry that it is applied to. Driven dimensions are automatically enclosed in parentheses to distinguish them from driving dimensions. You can add as many driven dimensions to the sketch as you wish.

Base Model

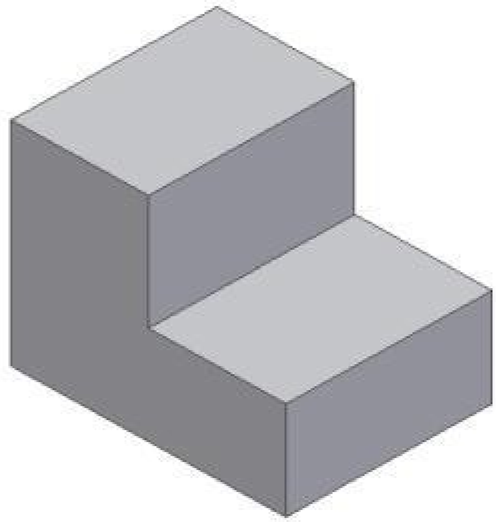
After the Base sketch is complete and is fully constrained, it is ready to be extruded or revolved to create the base model. The *Base model* is the solid model created from the base sketch by extruding or revolving it. In this module, only extruding the Base sketch to create the Base model is taught. The simplest definition of an extrusion is it adds depth to the Base sketch to create the Base model. See Figure 5-1.



The Base Sketch



The Extrusion



The Base Model
Figure 5-1

Inventor Command: EXTRUDE

The EXTRUDE command is used to extrude a 2D sketch to create or edit a 3D solid model.

Shortcut: **E**

Dimensioning Sketches

There are many different types of dimensions available to the operator to dimension sketches. The different dimensioning types will be taught throughout the Inventor book. In this module, inserting object linear dimensions is taught.

Linear Dimensions

A *linear* dimension is a dimension measuring the *delta X* or the *delta Y* distance between the two XY locations or endpoints of a line. See Figure 5-2. Linear dimensions are always either horizontal (delta

X) or vertical (delta Y). A linear dimension cannot be used to dimension the true length of an inclined line. It will only dimension the true length of a line if the line is horizontal or vertical. If both endpoints of a line lie on the same axis, it can only be dimensioned in one delta direction. Since all lines that were drawn to this point in the book were either horizontal or vertical, linear dimensions will be used for all of the dimensions inserted in this module.

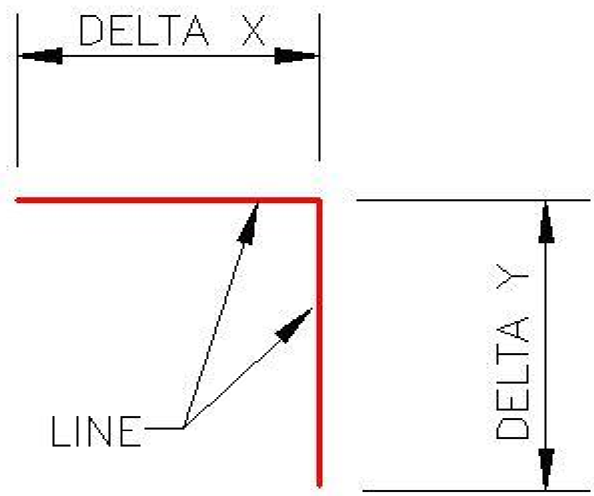


Fig 5-2
Linear Dimensions Applied to
Horizontal and Vertical Lines

Object Linear Dimensions

To insert an *object linear dimension*, enter the GENERAL DIMENSION command and move the cursor onto the line to be dimensioned. When the two headed arrow icon appears, click the left mouse button to select the line as shown in Figure 5-3. After selecting the line, move the cursor in the direction to select the location of the dimension. Click the mouse at the desired location of the dimension.

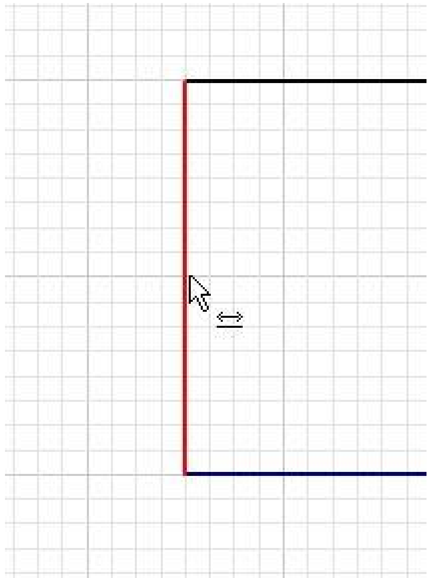
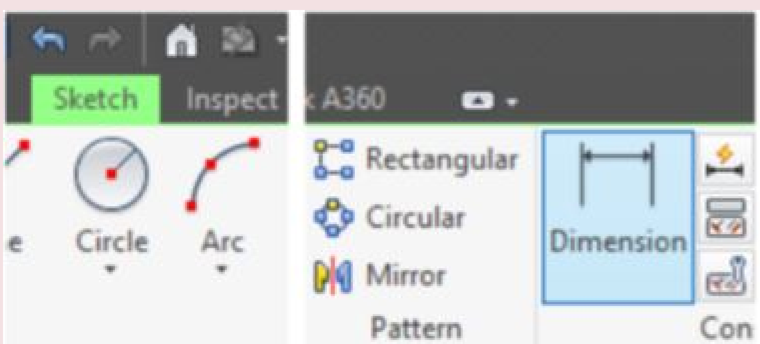


Figure 5-3 Object Linear Dimension Icon

Inventor Command: GENERAL DIMENSION

The GENERAL DIMENSION command is used to create driving or driven dimensions on a sketch.

Shortcut: **D**



WORK ALONG: Dimensioning and Extruding the Base Sketch

Step 1

Using the OPEN command, open the part file: Inventor Workalong 04-1 that you completed in Module 4.

Step 2

Change to Sketch mode by editing Sketch1 in the Browser bar. Press F6 to change the view to the Home view. (Figure Step 2)

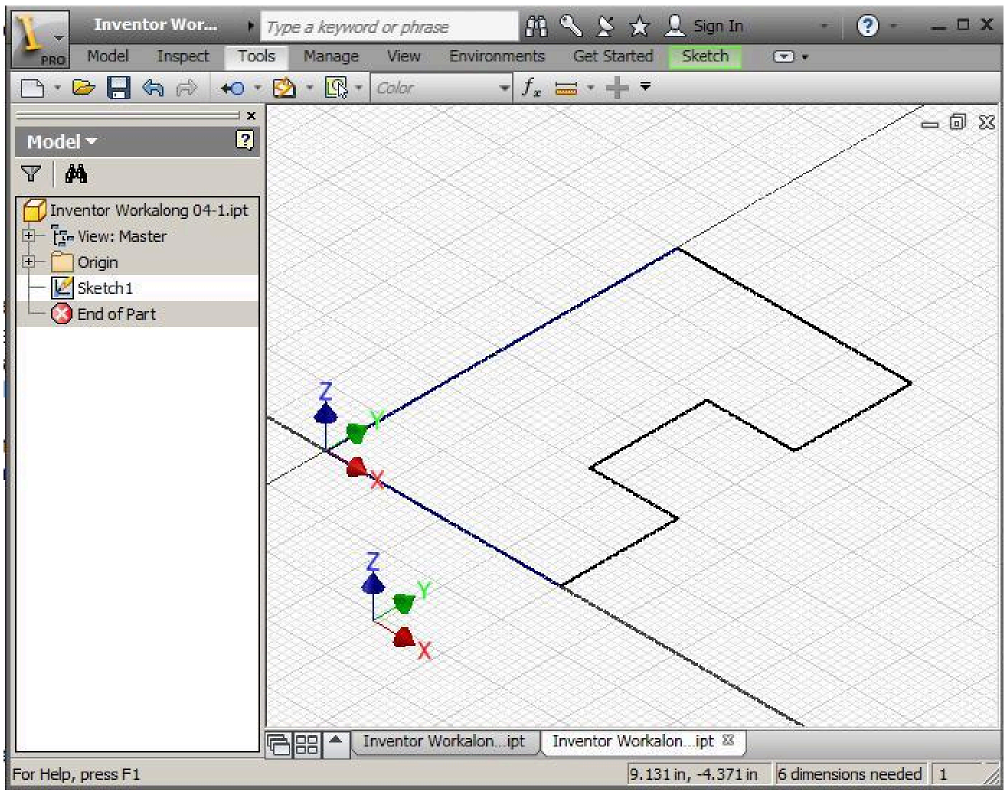
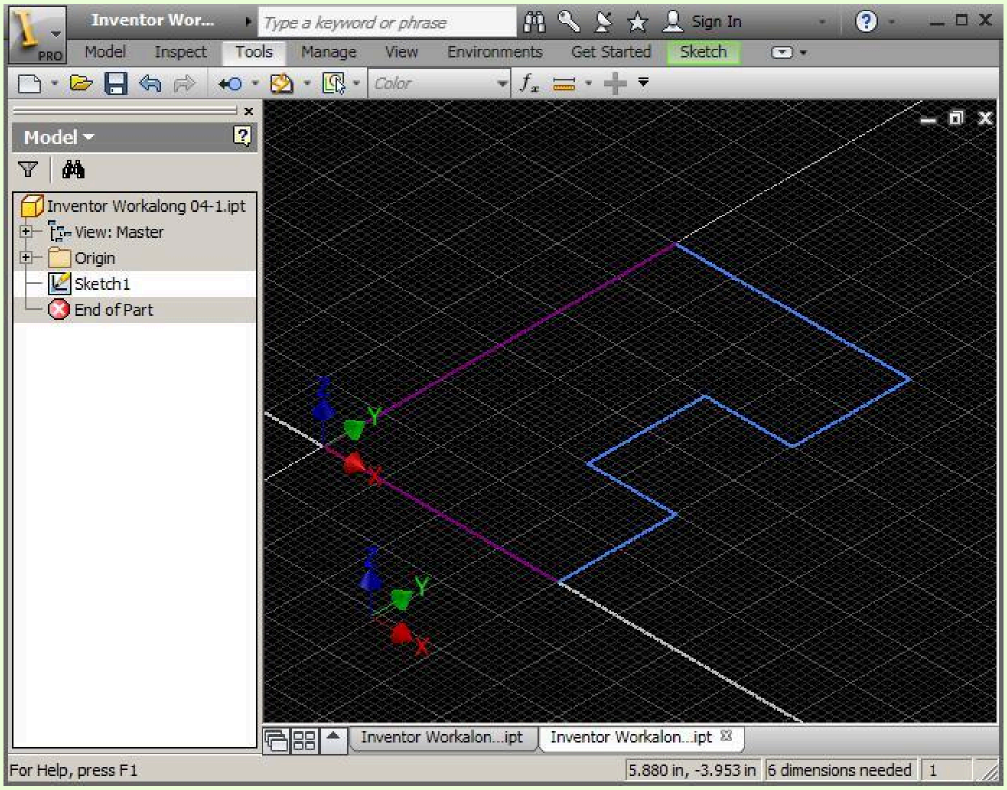


Figure Step 2

AUTHOR'S COMMENTS: Two lines of the sketch should appear purple and the remainder blue. The purple lines are constrained while the blue ones are not. The reason the two lines are constrained is because they have one endpoint snapped to the Center Point (X0Y0), which is constrained (projected) to the sketching plane. In this workalong, you will be adding dimensional constrains to fully constrain the sketch.



Step 3

Enter the LOOK AT command and when prompted, select one of the lines on the sketch as shown in the figure. (Figure Step 3A and 3B)

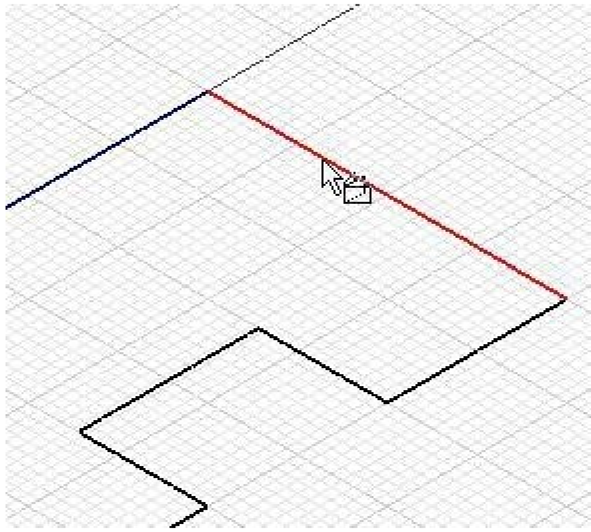


Figure Step 3A

AUTHOR’S COMMENTS: For the beginner, it is sometimes easier to dimension the sketch while it is in a 2D view. While this step is not necessary, I used it here to show you different methods of working on the sketch.

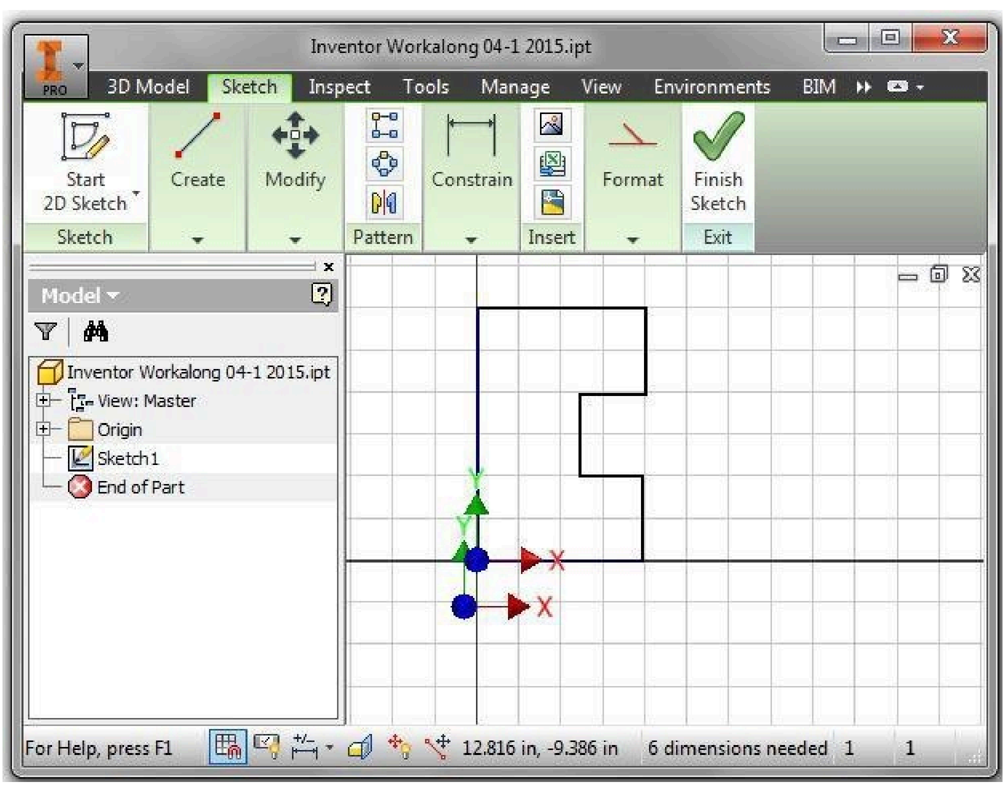


Figure Step 3B

Step 4

Enter the GENERAL DIMENSION command by pressing D on the keyboard. Move the cursor onto the left vertical line. When the Two Headed Arrow icon appears, select the line by clicking the left mouse button. (Figure Step 4)

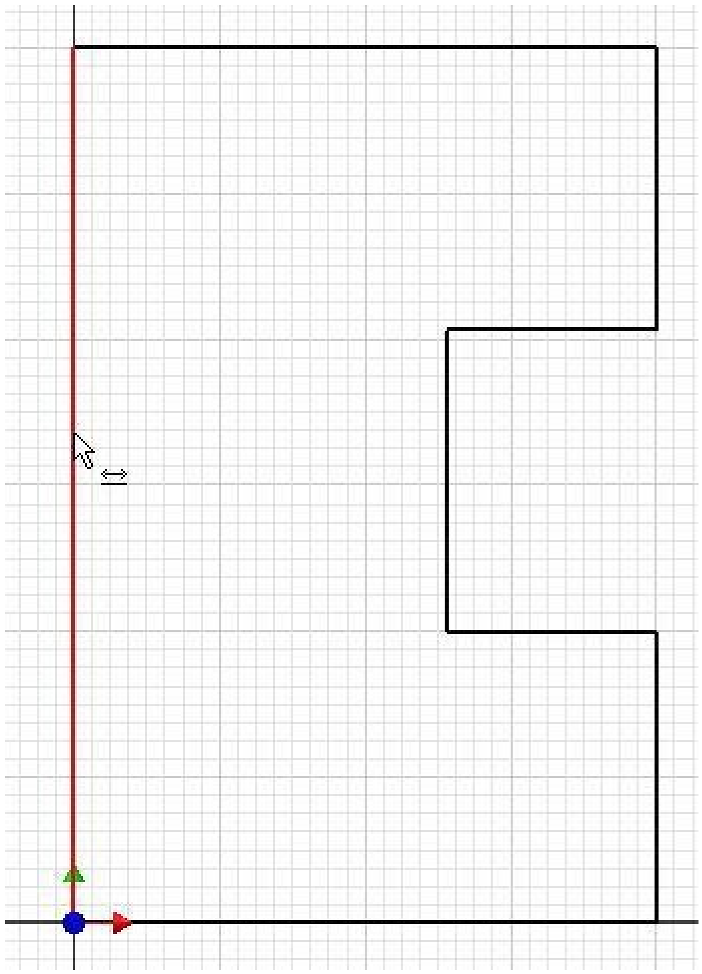


Figure Step 4

AUTHOR'S COMMENTS: It is best to dimension starting at lines that snapped to the Center Point (X0Y0). Create dimensioned in both directions moving around the sketch until the sketch is fully constrained.

Step 5

Move the cursor to the left and drag the dimension. Inventor will measure the delta Y length of the line, since it is a vertical line, and display its actual length. In this case, it is 6.0 inches long. Since that is the desired length, accept it by clicking the green Check icon. (Figure Step 5)

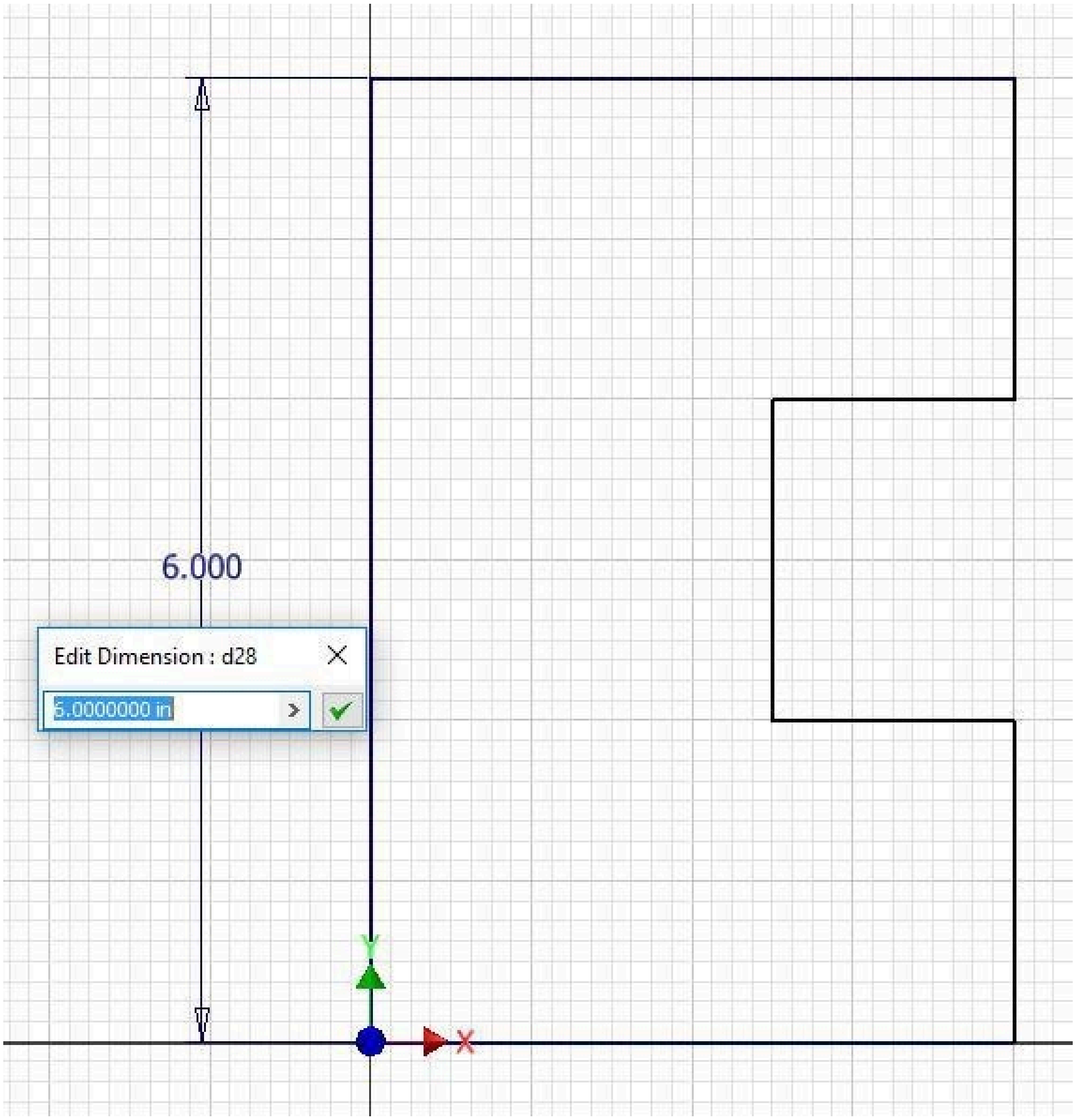


Figure Step 5

Step 6

Continue to add linear dimensions to fully constrain your sketch. If the length of the lines are correct, accept the dimension. If they are incorrect, change the dimension in the edit box to the actual dimension of the line and then accept it by clicking the green Check icon. (Figure Step 6A, 6B, 6C, and 6D)

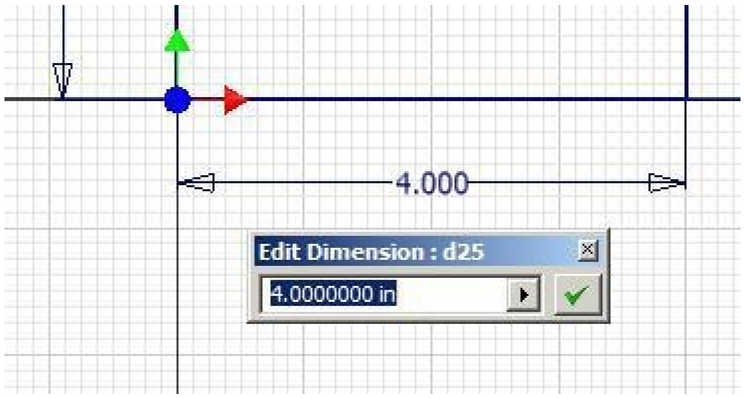


Figure Step 6A

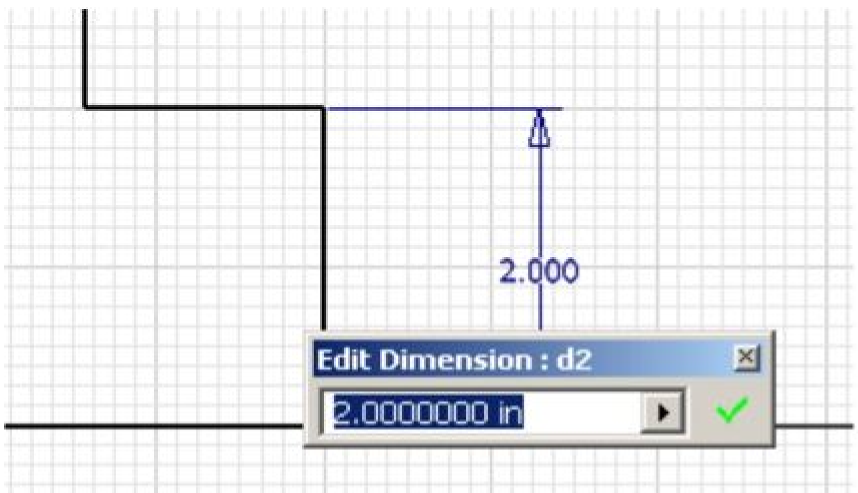


Figure Step 6B

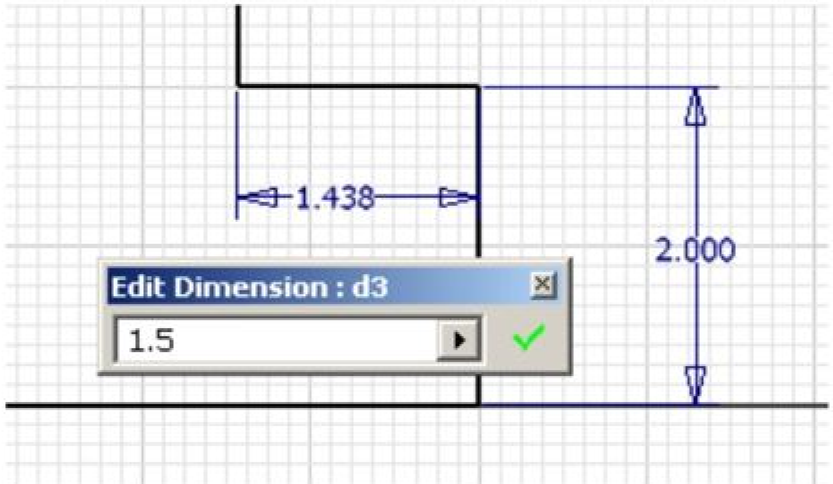


Figure Step 6C

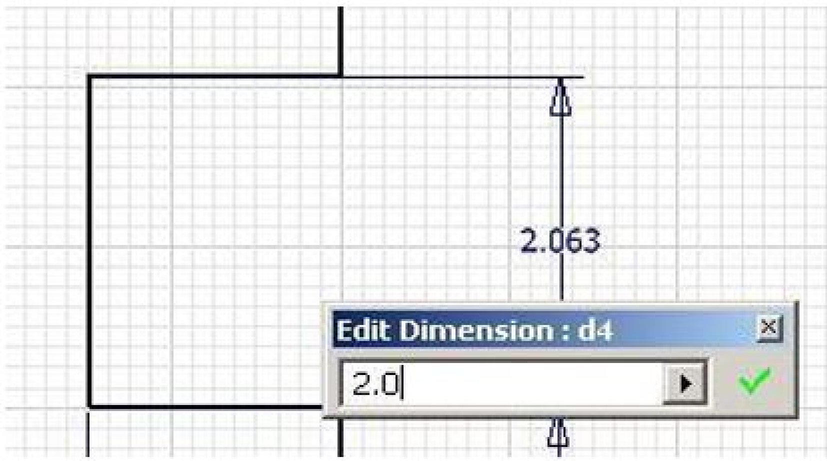


Figure Step 6D

Step 7

When complete, there should be six driving dimensions. (Figure Step 7)

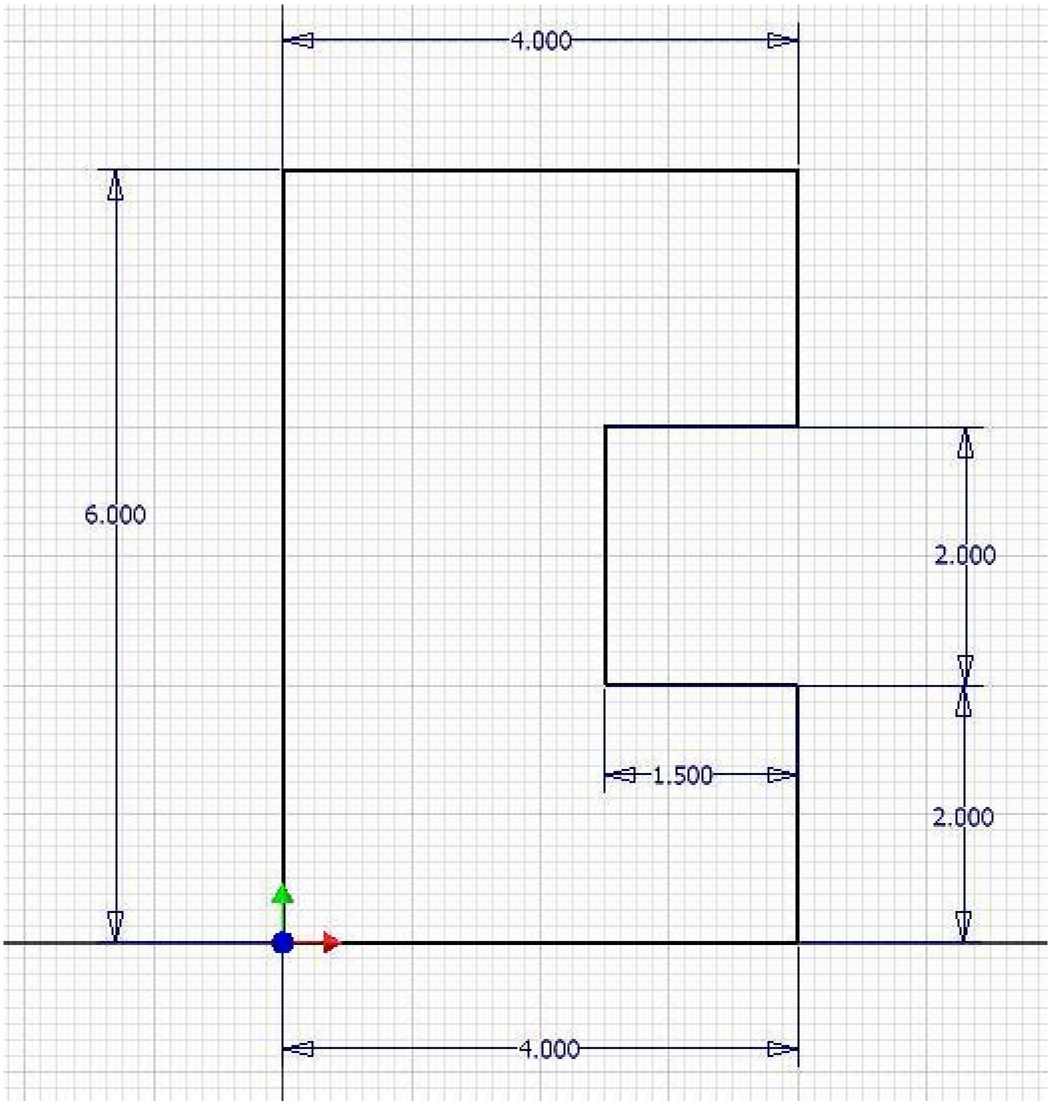


Figure Step 7

Step 8

Add one more dimension as shown in the figure. Note that Inventor will issue a warning that the dimension over-constrains the sketch. Click the Accept button to add it as a driven dimension. (Figure Step 8A and 8B)

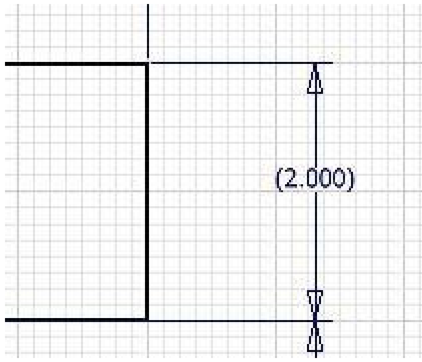


Figure Step 8A

AUTHOR'S COMMENTS: This step is not necessary to fully constrain the sketch. Driven dimensions are added for reference only.

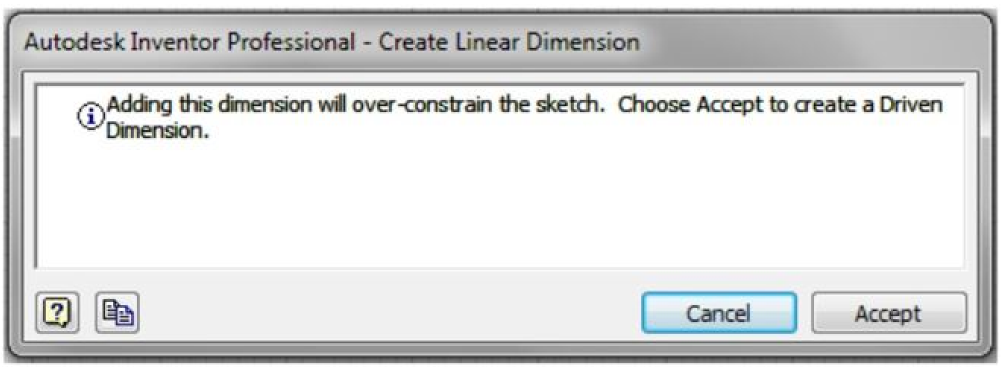


Figure Step 8B

Step 9

Without entering a command, move the cursor onto the 2.000 dimension as shown in the figure. When the Move icon displays, click and hold the mouse button down and drag the dimension to the new location shown in the figure. (Figure Step 9A and 9B)

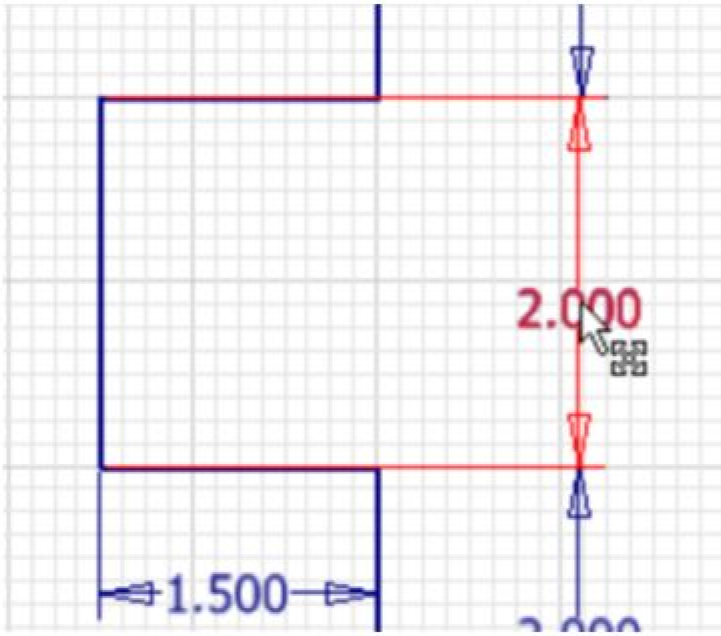


Figure Step 9A

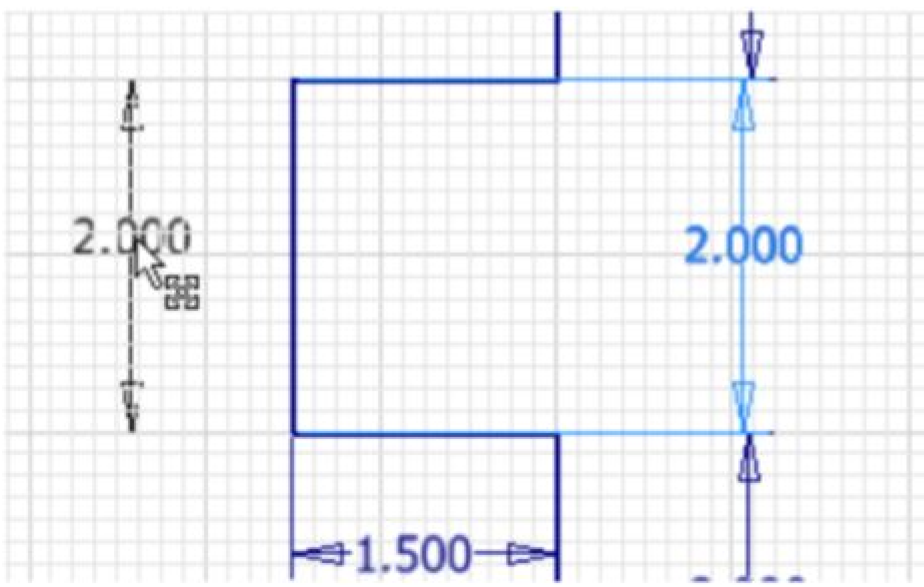


Figure Step 9B

AUTHOR'S COMMENTS: I use the Move icon to centre the numbers in the dimensions after I complete the sketch. This is not a necessary step but makes the sketch appear more professional.

USER TIP: Instead of clicking the green Check icon to accept the displayed dimension in the Edit Dimension dialogue box it is faster to press the Enter key.

Step 10

The completed sketch should appear as shown in the figures. (Figure Step 10A, 10B, and 10C)

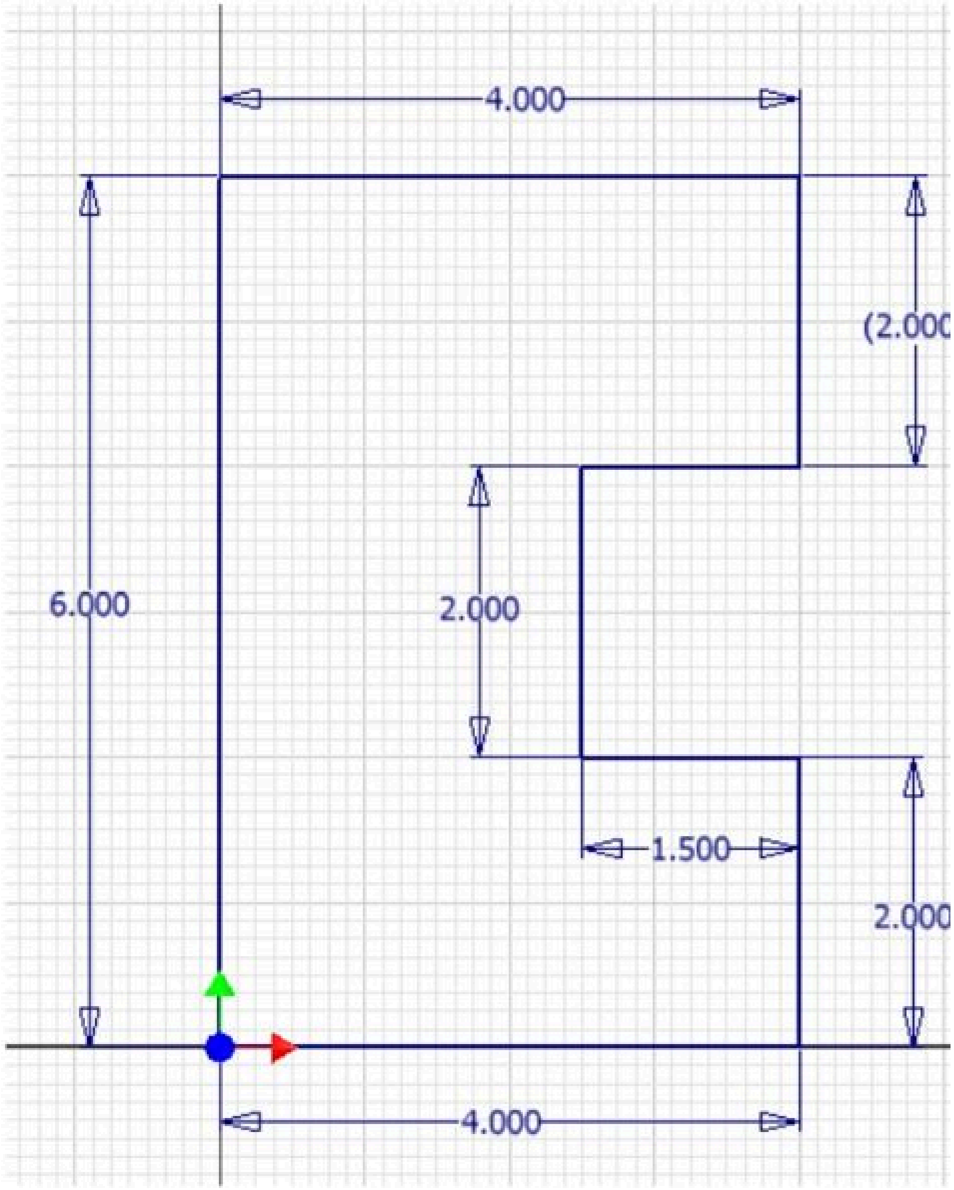


Figure Step 10A

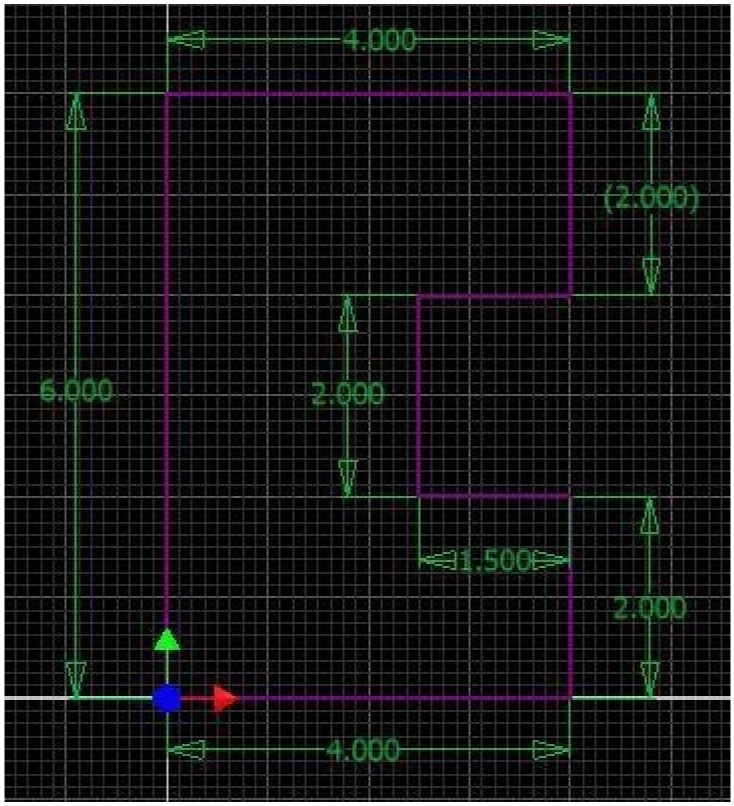


Figure Step 10B

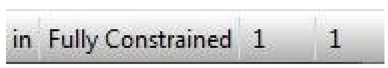


Figure Step 10C

AUTHOR'S COMMENTS: All of the lines in your sketch should appear purple and the Status bar should display Fully Constrained as shown in Figure Step 10C.

Step 11

Press F6 to change to the Home view. (Figure Step 11)

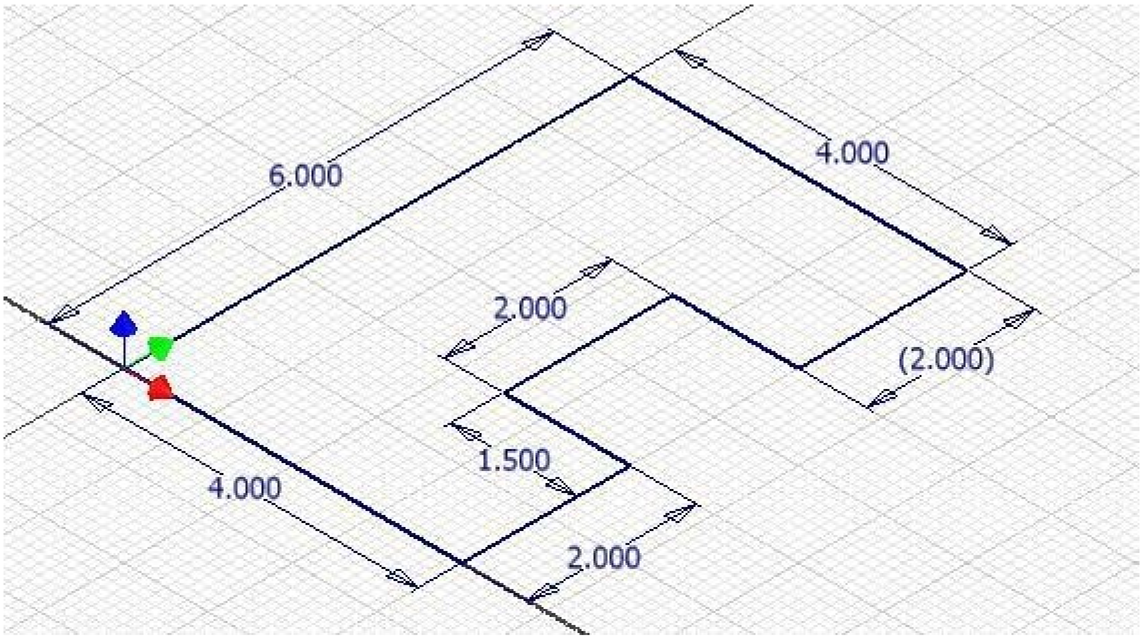


Figure Step 11

Step 12

Click the LOOK AT command and change the view to the top or XY plane.

Step 13

Press F8 to display the geometrical constraint icons. You figure should appears similar as shown in the figure. (Figure Step 13)

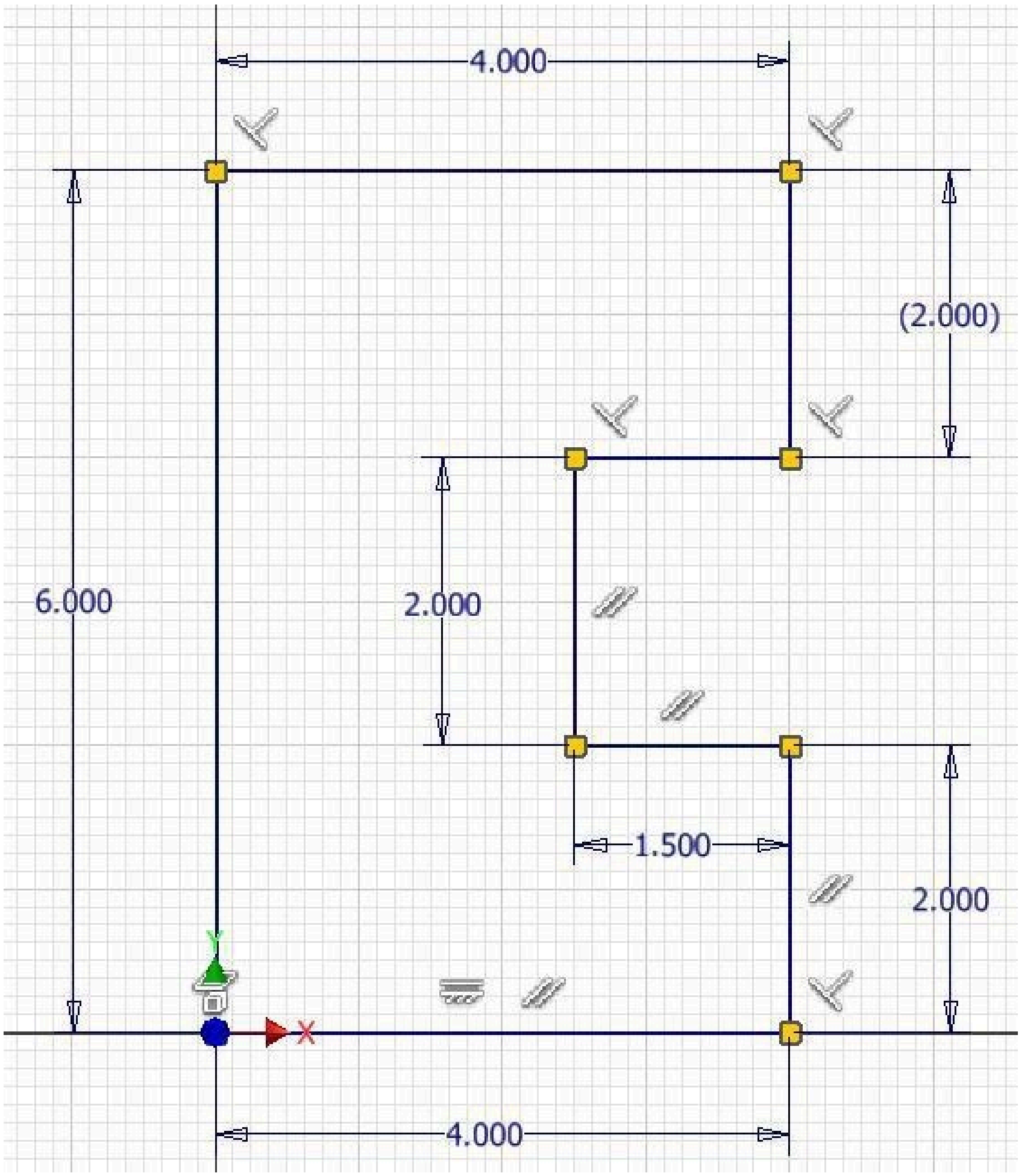


Figure Step 13

AUTHOR'S COMMENTS: Your geometrical constrains may not match the figure exactly. Just ensure that the sketch is fully constrained.

Step 14

Enter the EXTRUDE command. The Extrude dialogue box will display. The model being drawn is 5 inches high as shown in the figure. Set the Output box to Solid, the Extents to Distance of 5 and the Extrude Direction to positive Z as shown in the figures. (Figure Step 14A and 14B)

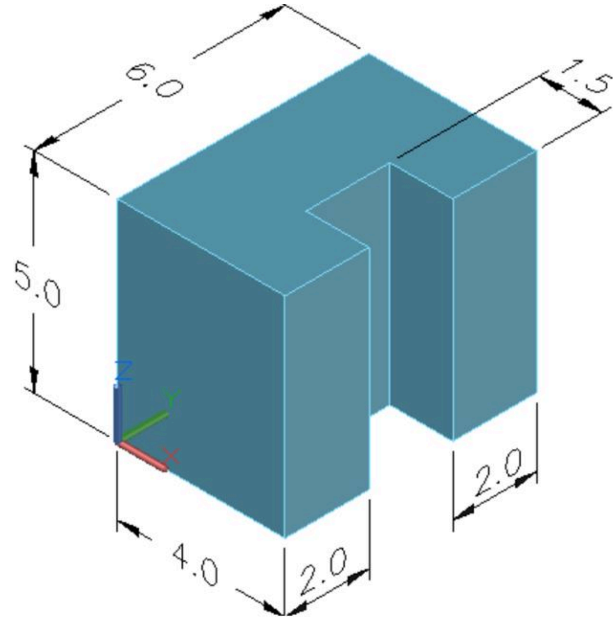


Figure Step 14A

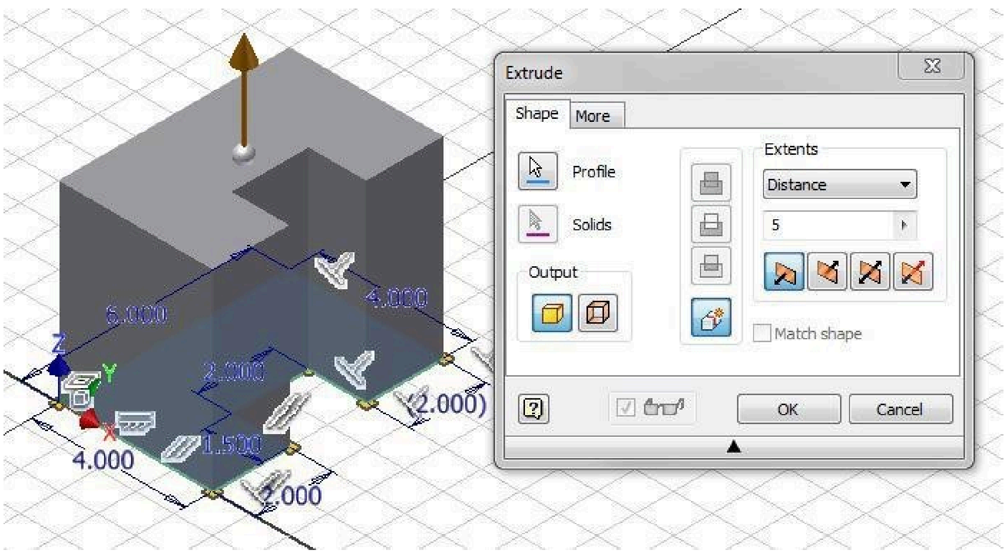


Figure Step 14B [Click to see image full size]

AUTHOR'S COMMENTS: Note that Inventor will, by default, attempt to extrude in the positive Z direction. This extrude direction can be reversed, when required.

Step 15

The completed solid model should now appear as shown in the figure. (Figure Step 15)

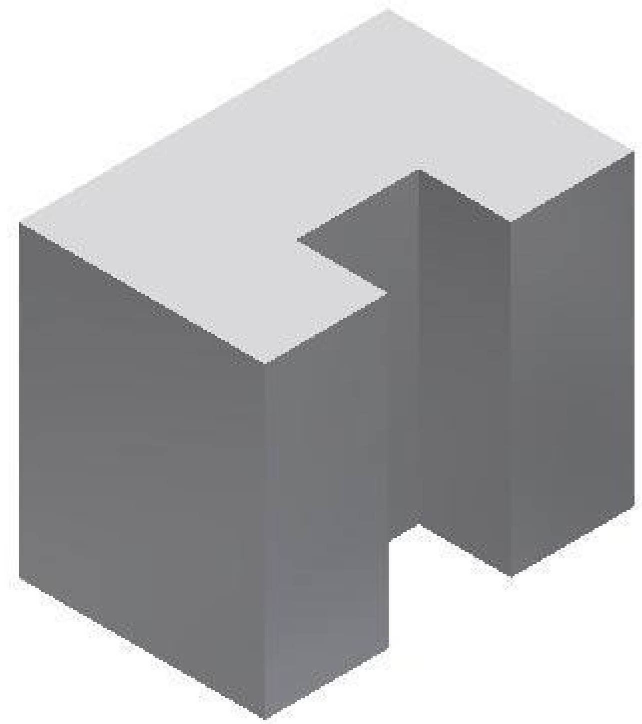


Figure Step 15

Step 16

Find the Appearance pull-down menu and select the arrow to pull down the Color list. Ensure that the library: Inventor Material Library is enabled at the bottom of the list. (Figure Step 16A and 16B)

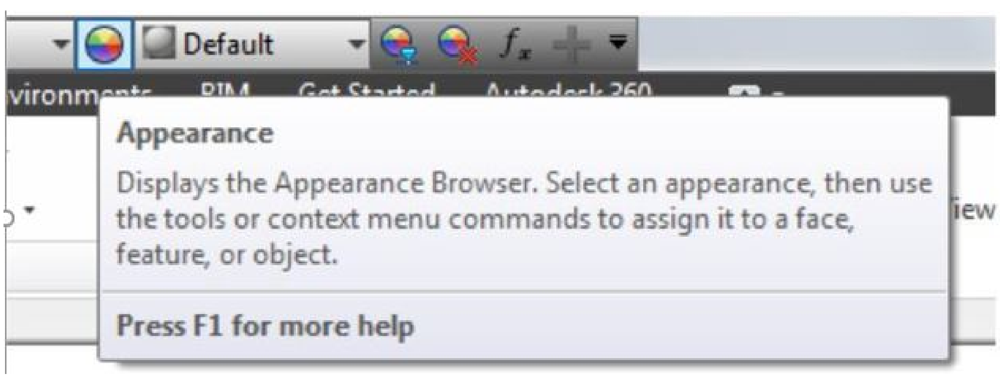


Figure Step 16A

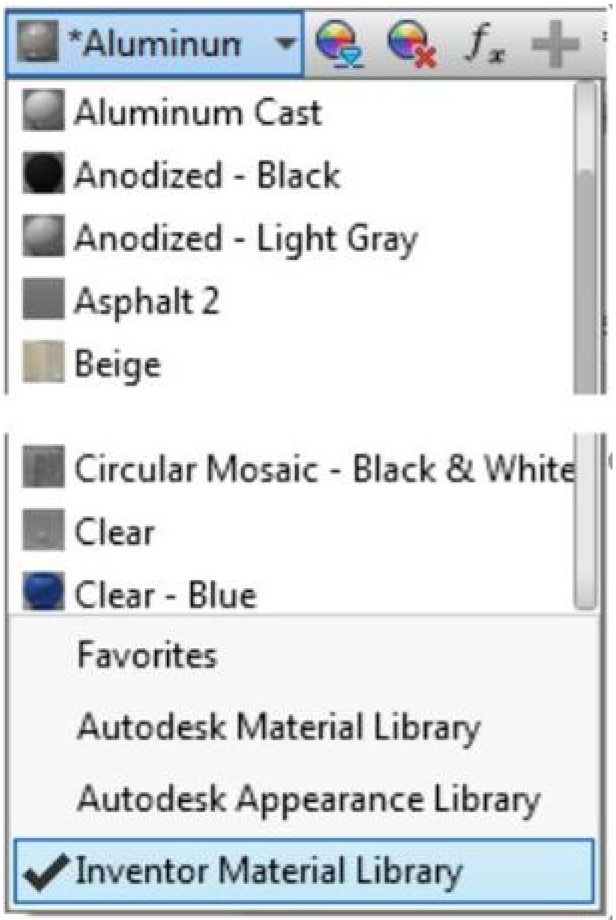


Figure Step 16B

Step 17

In the Appearance pull-down menu, select the color: Aluminum Polished as shown in the figure. (Figure Step 17)

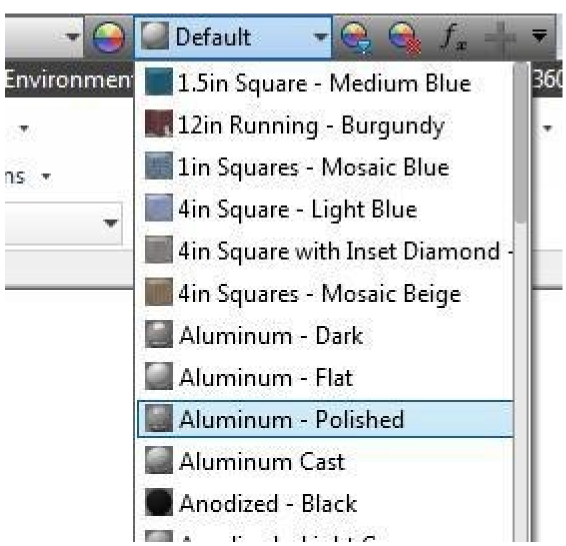


Figure Step 17

AUTHOR'S COMMENTS: Ensure that you use the Appearance pull-down list and not the Material pull-down **Figure Step 17** list to the left of the Appearance pull-down list. The Material pull-down list will be covered in Module 20.

AUTHOR'S COMMENTS: If your Inventor software does not include Inventor Material Library, you can use either the Autodesk Material Library or the Autodesk Appearance Library to set the colors specified in the book.

Step 18

The completed part or model should appear as shown in the figure. (Figure Step 18)

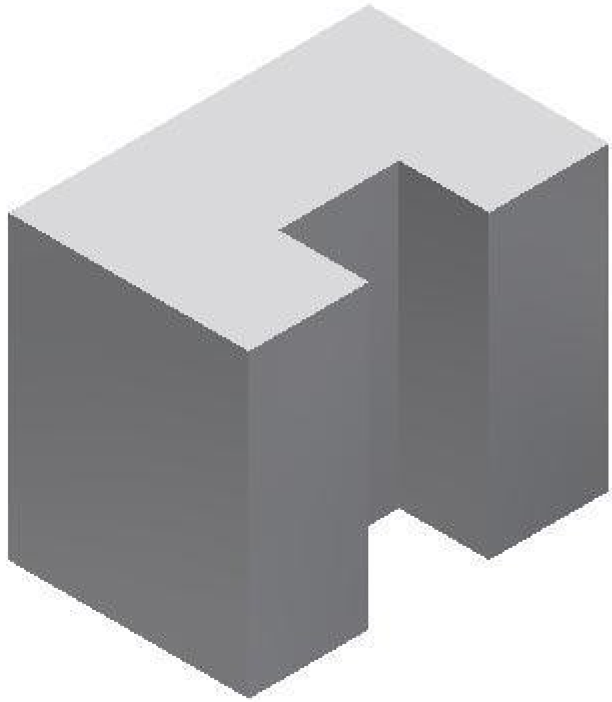


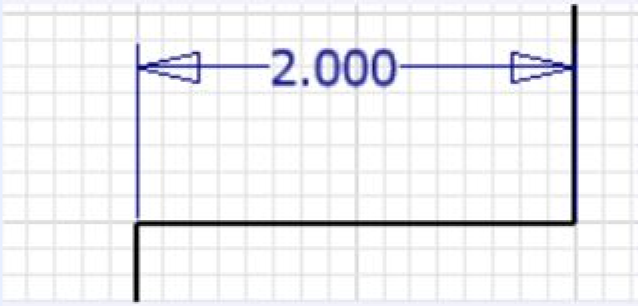
Figure Step 18

AUTHOR'S COMMENTS: The colour only changes the model to appear as the assigned colour. It does not specify the material that the solid model is made from. That will be taught in Module 20.

Step 19

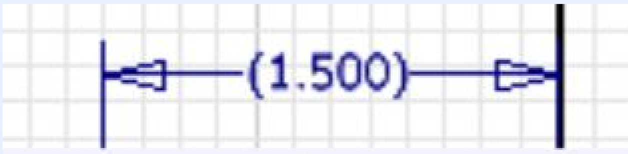
Save and close the part file.

MUST KNOW: A driving dimension is a parametric dimension controlling the size of the object. Inventor will automatically change the overall sketch to conform to the driving dimensions maintaining the existing geometrical constraints that were used in the design.



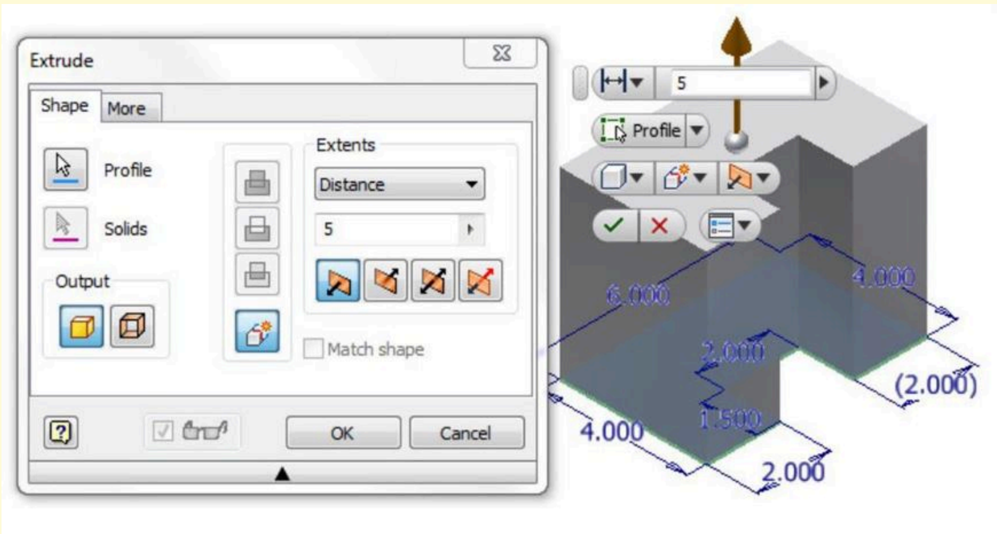
Driving Dimension

A driving dimension is shown below. A driven dimension is a non-parametric dimension that does not constrain the object. It is inserted for reference only and is always displayed enclosed in brackets as shown below.



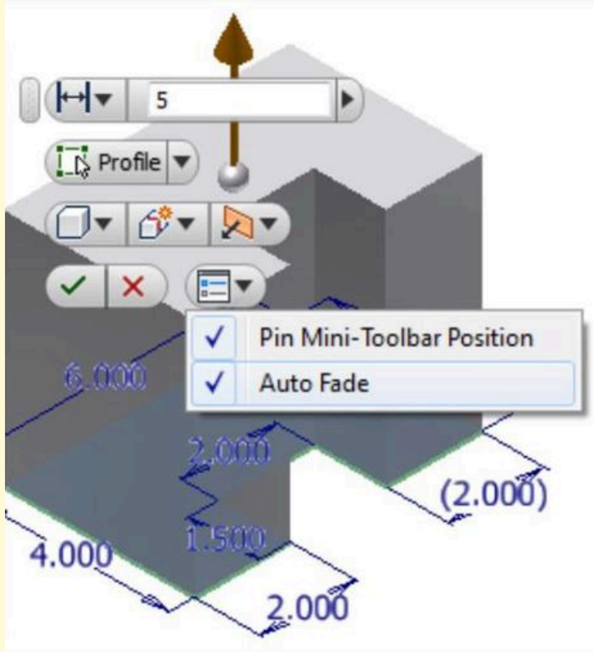
Driven Dimension

USER TIP: When you open the Extrude dialogue box for the first time, the Marking menu will display on top of the extruded model. You will not be using that menu in the Inventor book.

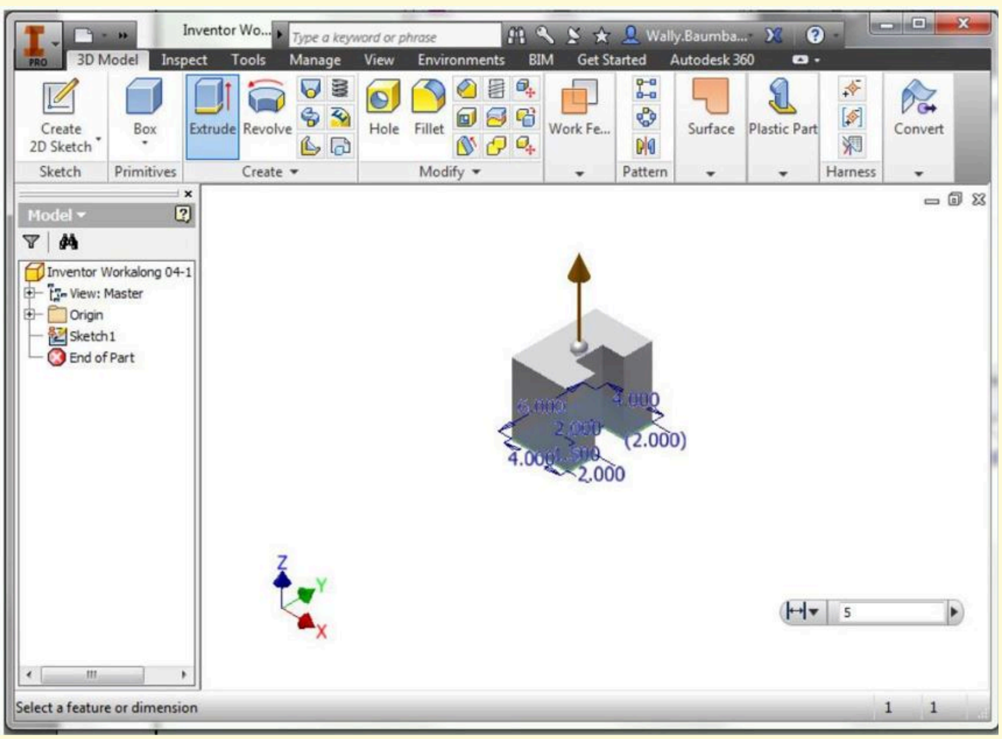


[Click to see image full size]

USER TIP: Click the bottom right arrow and enable Pin Mini-Toolbar Position and Auto Fade.

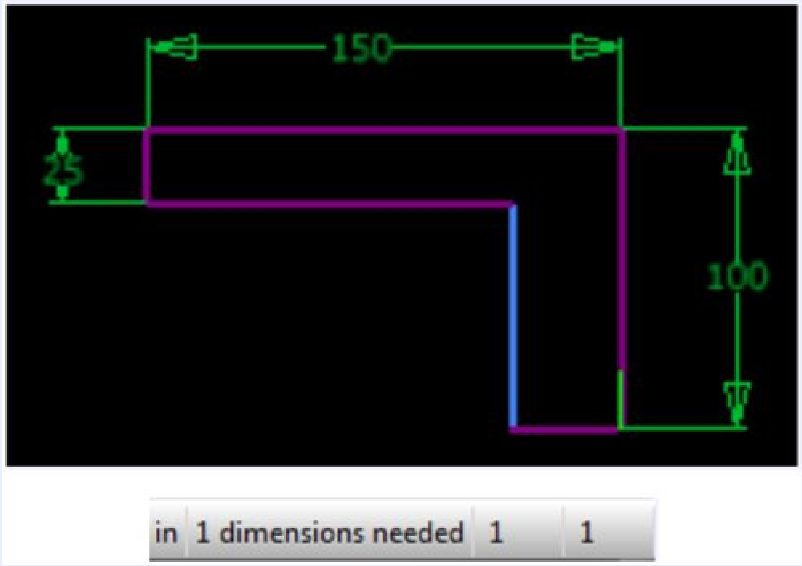


Drag the Marking menu to bottom right corner of the Graphic window. It will remain there for the duration while you working on the Inventor book. After you complete the book, you can use change the settings to use the menu any way the works for you.

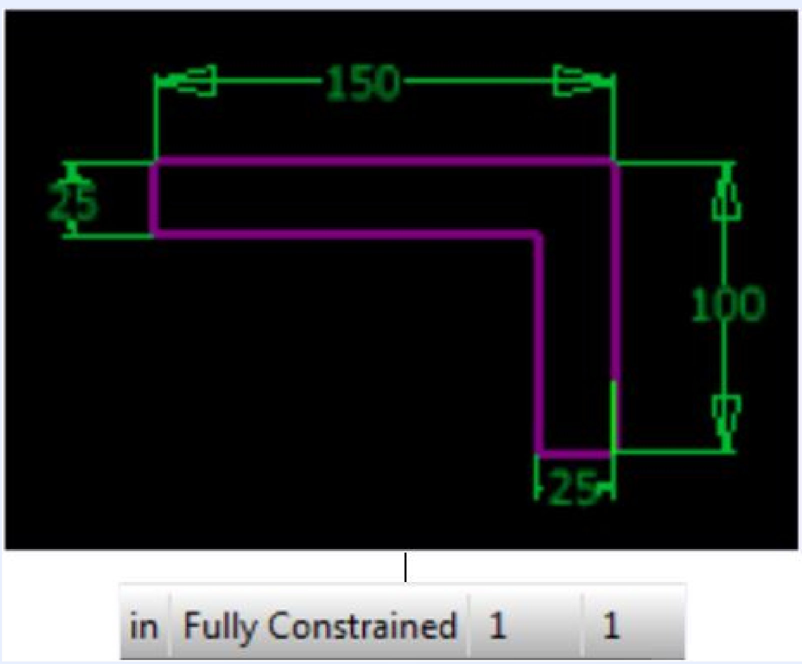


[Click to see image full size]

MUST KNOW: **Knowing When the Sketch is Fully Constrained** It is very important to know when a sketch is fully constrained. A fully constrained sketch is complete and is ready to be extruded or revolved to create or edit the solid model. When the colour scheme is set to High Contrast, the background is black and the lines that are constrained will display purple. The lines that are not yet constrained will display blue. When the sketch is fully constrained, all of the lines in the sketch will display purple. Inventor also reports the current constraint status on the Status bar. See the figures below.

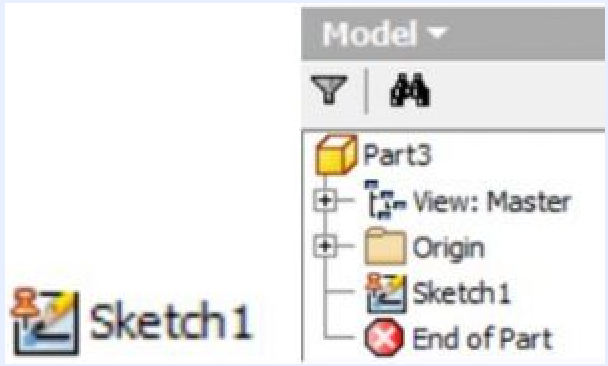


Sketch Partially Constrained



Sketch Fully Constrained

A special Pinned icon will display in Browser bar on sketches that are fully constrained as shown to the figures.



Key Principles

Key Principles in Module 5

1. Unlike geometrical constraints, that are used to apply geometrical relationships of the objects in a 2D sketch, dimensional constraints control and report the size of the geometry.
2. To fully constrain a 2D sketch, driving dimensions must be applied. A driving dimension is a parametric dimension controlling the size of the object. Inventor will automatically change the overall object to conform to the driving dimensions maintaining the existing geometrical constraints that were assigned in the sketch.
3. A driven dimension is a non-parametric dimension that does not constrain the sketch.
4. It is very important to know when the sketch is fully When it is fully constrained, the sketch is complete and it is ready to be extruded or revolved to create the Base model. When the background colour scheme is set to High Contrast the constrained lines will display purple and lines that are not yet constrained will display blue. When the sketch is fully constrained, all of the lines in the sketch will display purple.
5. After the Base sketch is complete and is fully constrained, it is ready to be extruded to create the Base model. The Base model is the solid model created from the Base sketch.

Lab Exercise 5-1

Time allowed: 40 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab 04-1	Inventor Course	Inches	English-Modules Part (in).ipt	Zinc Chromate 2	N/A

Step 1

Open the file: Inventor Lab 04-1.ipt that you saved in Lab Exercise 4-1 in Module 4.

Step 2

Using the SAVEAS command, save the file with the name: Inventor Lab 05-1.ipt.

Step 3

Insert the necessary driving dimensions to fully constrain the sketch. Add at least one driven dimension. (Figure Step 3A, 3B, and 3C)

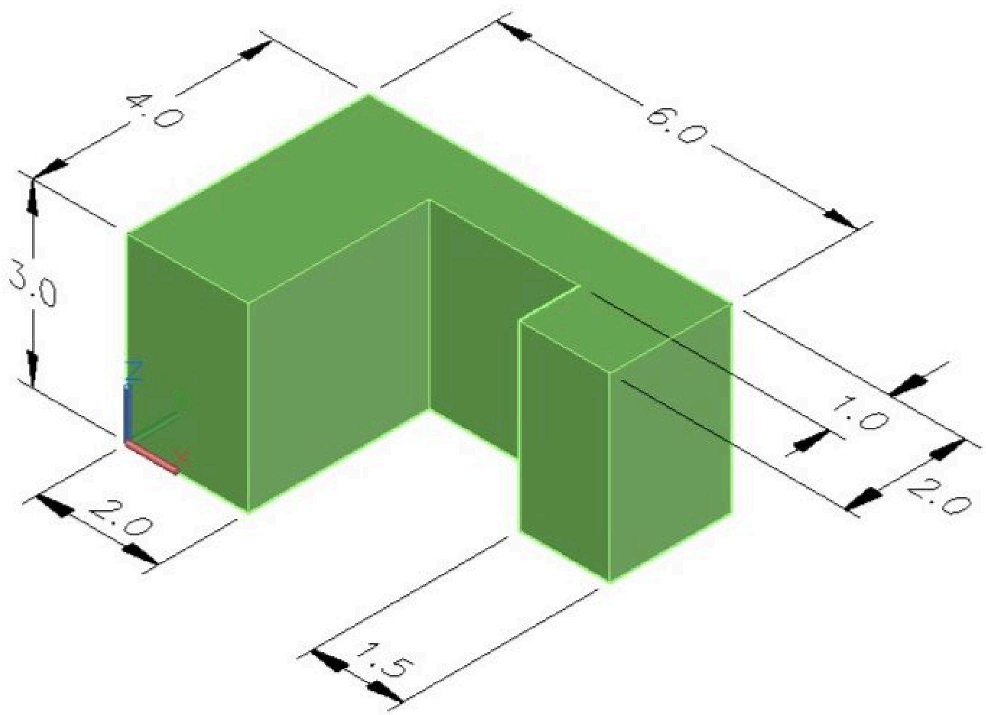


Figure Step 3A
3D Model

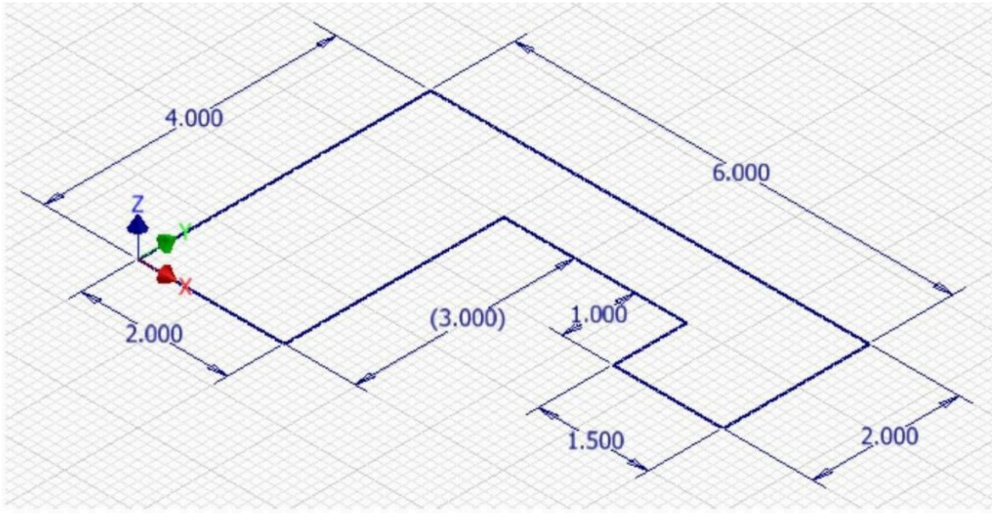


Figure Step 3B
Base Sketch Dimensioned

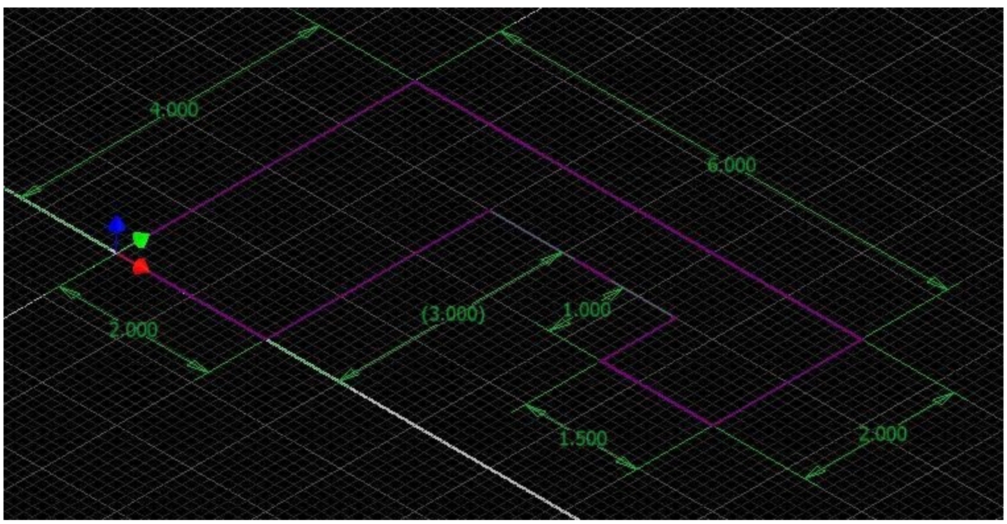


Figure Step 3C
The Base Sketch Fully Constrained [Click to see image full size]

Step 4

Extrude the sketch to create the Base model and apply the colour shown above. (Figure Step 4)

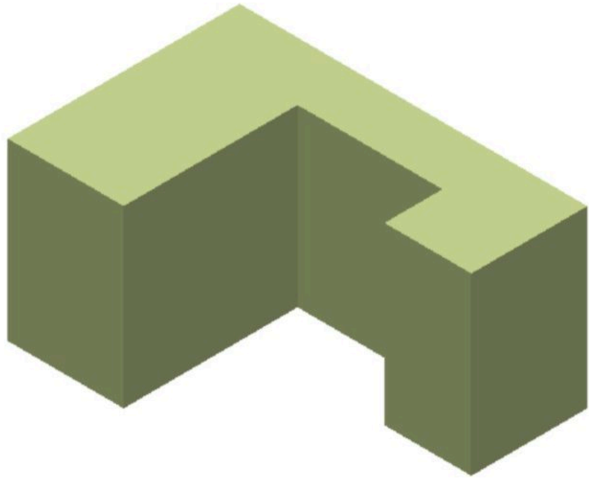
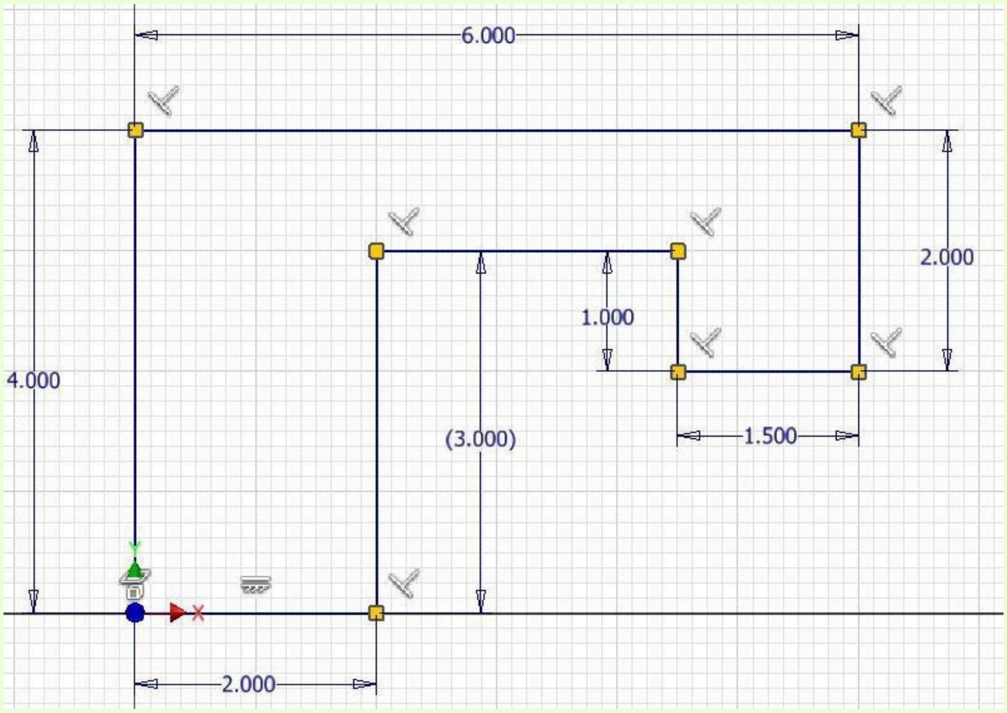


Figure Step 4 Completed 3D Solid Model Home View

AUTHOR'S GEOMETRIC CONSTRAINTS: The following figure shows the Base sketch's construction method plus geometric and dimensional constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketch using a different construction method and constraints, that is what is important. You may want to compare your construction method and constraints used with the authors.



[Click to see image full size]

Lab Exercise 5-2

Time allowed: 40 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab 04-2	Inventor Course	Inches	N/A	Chrome – Polished Blue	N/A

Step 1

Open the file: Inventor Lab 04-2.ipt that you saved in Lab Exercise 4-2 in Module 4.

Step 2

Using the SAVEAS command, save the file with the name: Inventor Lab 05-2.ipt.

Step 3

Insert the necessary driving dimensions to fully constrain the sketch and add at least two driven dimensions. (Figure Step 3A, 3B, and 3C)

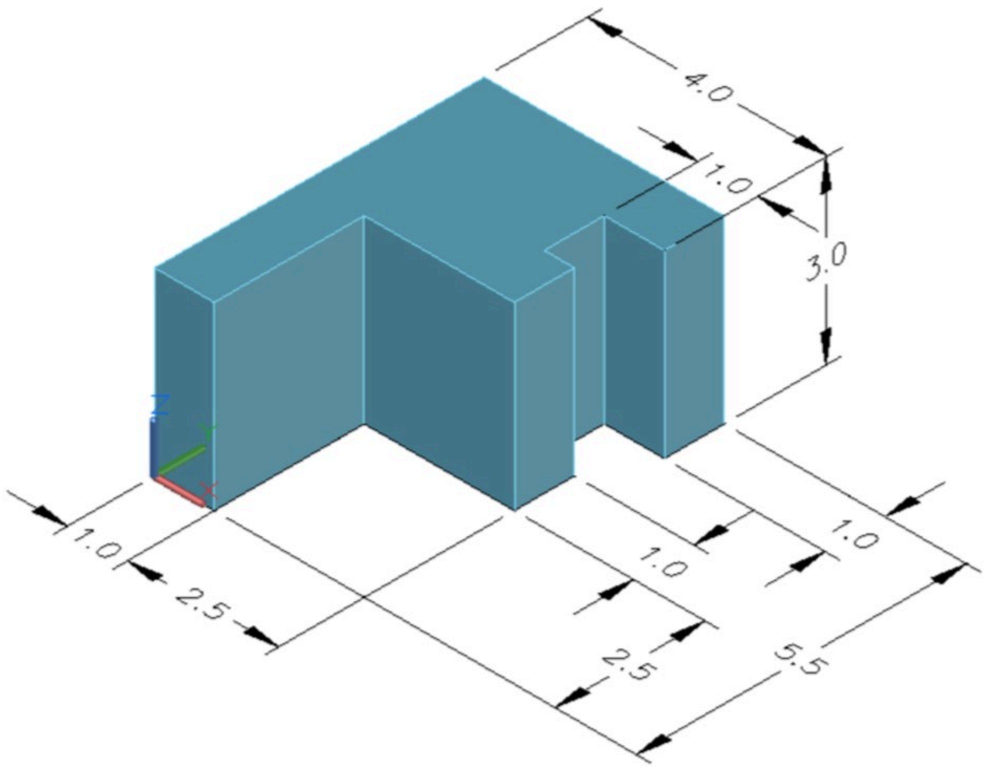


Figure Step 3A
3D Mode

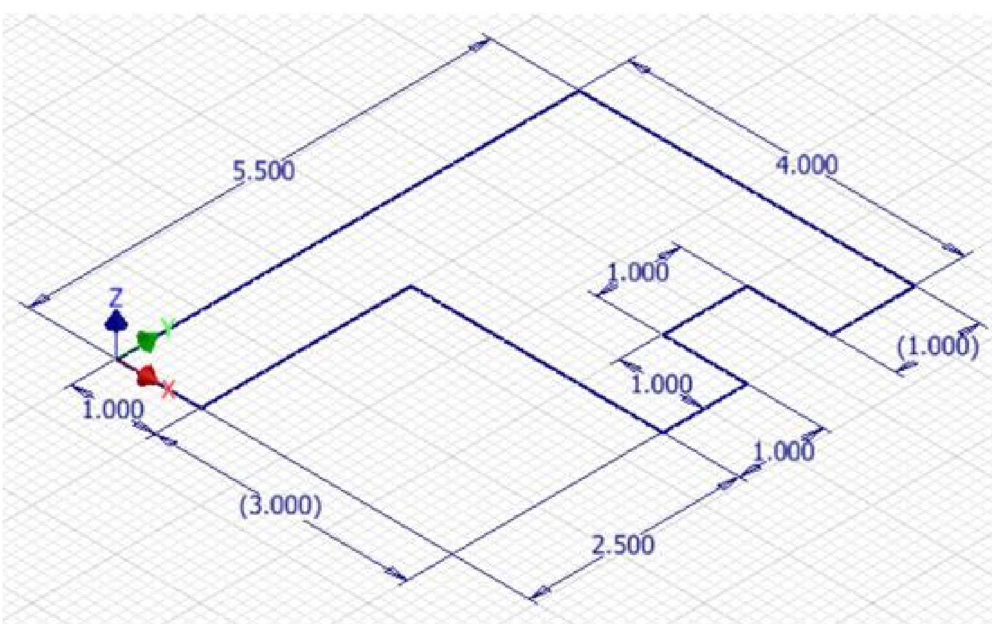


Figure Step 3B
Base Sketch Dimensioned [Click to see image full size]

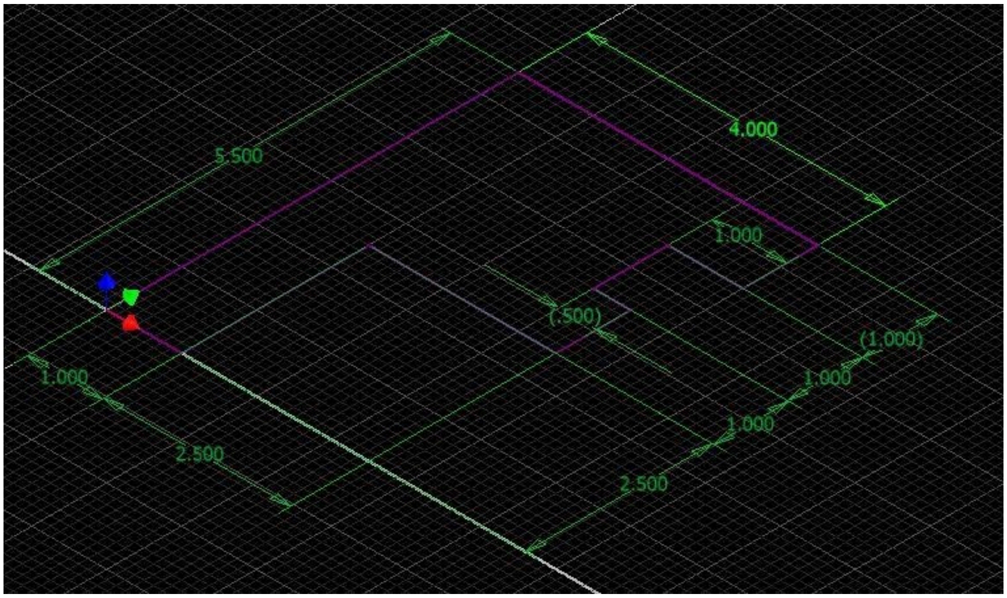


Figure Step 3C
Base Sketch Fully Constrained [Click to see image full size]

Step 4

Extrude the sketch to create the Base model and apply the colour shown above. (Figure Step 4)

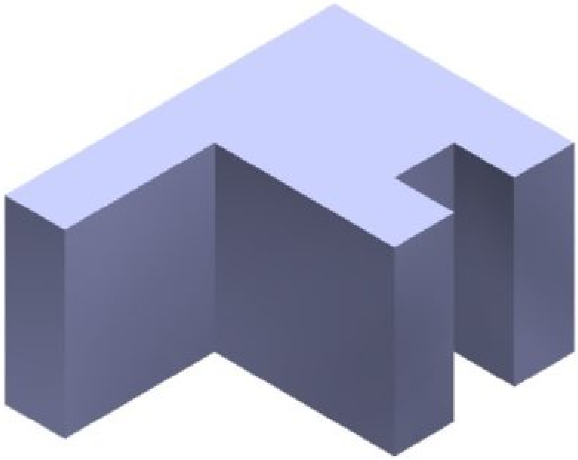
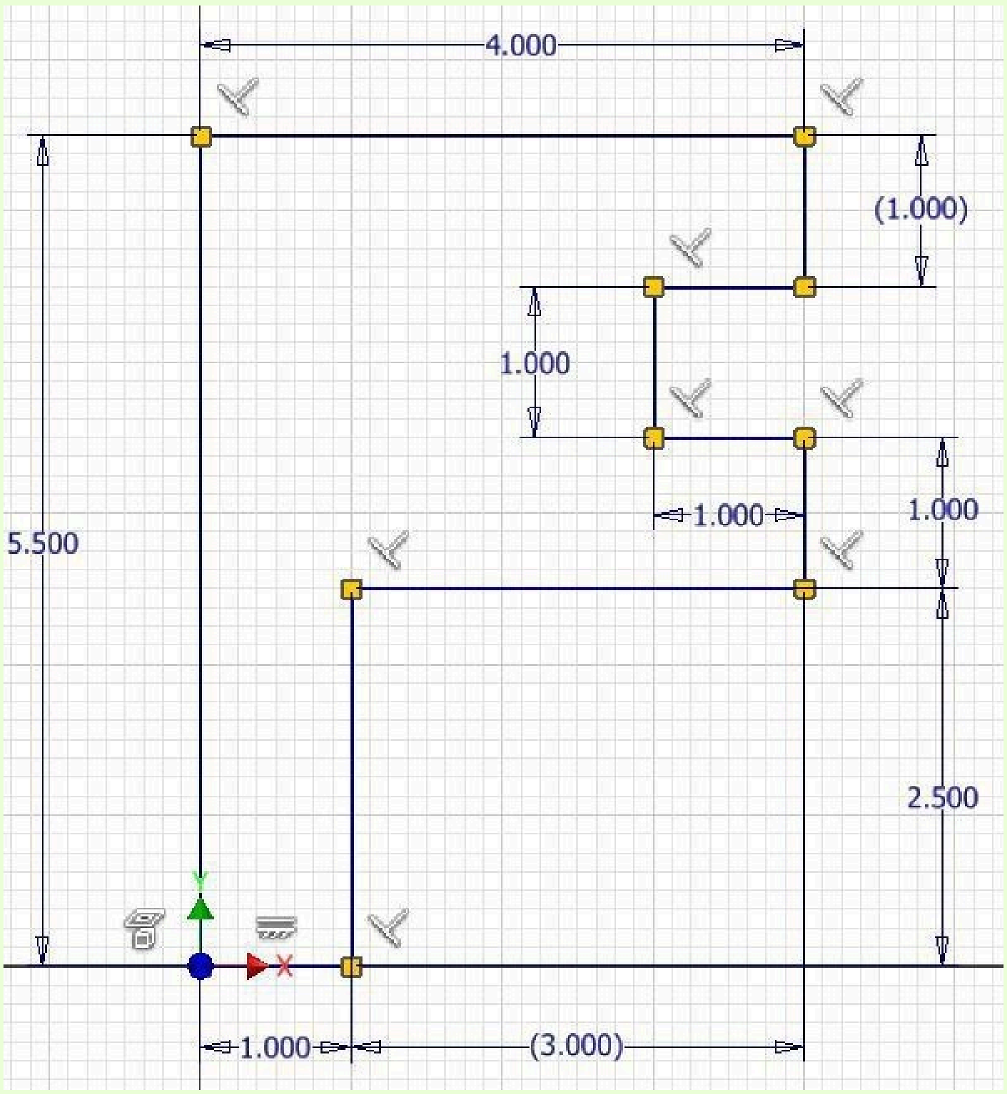


Figure Step 4
Completed 3D Solid Model
Home View

AUTHOR'S GEOMETRIC CONSTRAINS: The following figure shows the Base sketch's construction method plus geometric and dimensional constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketch using a different construction method and constraints, that is what is important. You may want to compare your construction method and constraints used with the authors.



[Click to see image full size]

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 hour time limit, complete a written exam and a lab exercise.

The Inventor 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 you have completed this module.

Multiple Choice Questions

Select the BEST answer.

1. What keyboard key is used to end the current command?
   A. CTRL
   B. TAB
   C. ESC
   D. SHIFT
   E. ENTER
2. An Inventor ________________is designed to logically organize, store and manage the valid links to the files that are created for each undertaking.
   A. 3D Model
   B. Project

C. Design

D. 2D Sketch

E. Menu

3. What command is used to zoom the model or the 2D sketch to display the complete model or sketch in the graphic window?

    A. ZOOM

    B. PAN

    C. ISOMETRIC

    D. ZOOM ALL

    E. LOOK AT

4. What are all files created in Inventor called?

    A. Designs

    B. Drawings

    C. Files

    D. 3D Models

    E. 2D Sketches

5. What command is used to change the viewing position of the model or sketch to a known home view or isometric view?

    A. ZOOM

    B. PAN

    C. HOME VIEW

    D. ZOOM ALL

    E. LOOK AT

6. What is the name of the Inventor file that must be used by the operator when creating any new file?

    A. Design file

    B. Drawing file

    C. Model file

    D. Template file

    E. Sketch file

7. What term describes the process of applying geometrical relationships of objects to one another in a 2D sketch?

    A. Object snapping

    B. Dimensional constraints

C. Parametric solid modeling

D. Driven dimensions

E. Geometrical constraints

8. What file extension is assigned to a part file?

A. .IPT

B. .IJP

C. .IAM

D. .IDW

E. .IPN

9. When drawing a base sketch that must be extruded to create the solid model there is a list of rules that should be used. Which one of the following rules is false?

A. Select the view with the most complex contour shape.

B. Draw the lines using dimensions close to finished dimensions.

C. The objects in the sketch must meet at their endpoints and cannot overlap.

D. The objects in the sketch must have geometrical constraints applied to control the shape.

E. The objects in the sketch cannot be a closed polygon.

10. What term describes the process of controlling and reporting the size of the geometry in a 2D sketch?

A. Object snapping

B. Dimensional constraints

C. Parametric solid modeling

D. Driven dimensions

E. Geometrical constraints

Lab Exercise 6-1

Time allowed: 60 minutes.

Part Name	**Project**	**Units**	**Template**	**Color**	**Material**
Inventor Lab Lab 06-1	Inventor Course	Inches	English-Modules Part (in).ipt	Aluminum – Flat	N/A

Step 1

Start a new part file with the template: English- Modules Part(in).ipt, and save the file with the name: Inventor Lab 06-1, as shown above.

Step 2

Project the Center Point onto the Base sketch.

Step 3

Draw the Base sketch for the object shown in the figure and apply all of the necessary geometrical constraints to maintain the shape of the sketch. Note the location of X0Y0Z0. (Figures Step 3A and 3B)

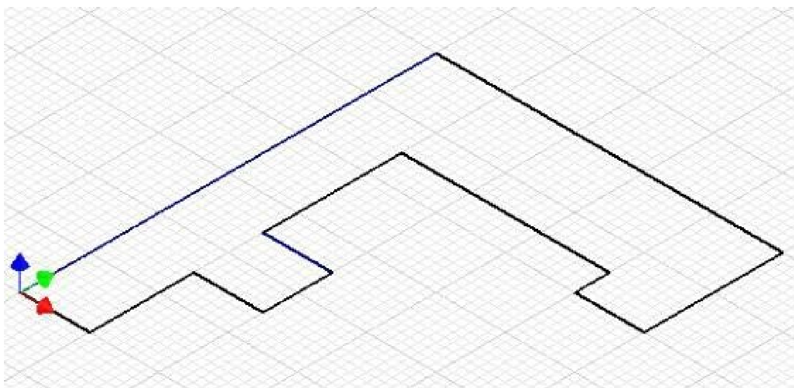


Figure 3A
Base Sketch [Click to see image full size]

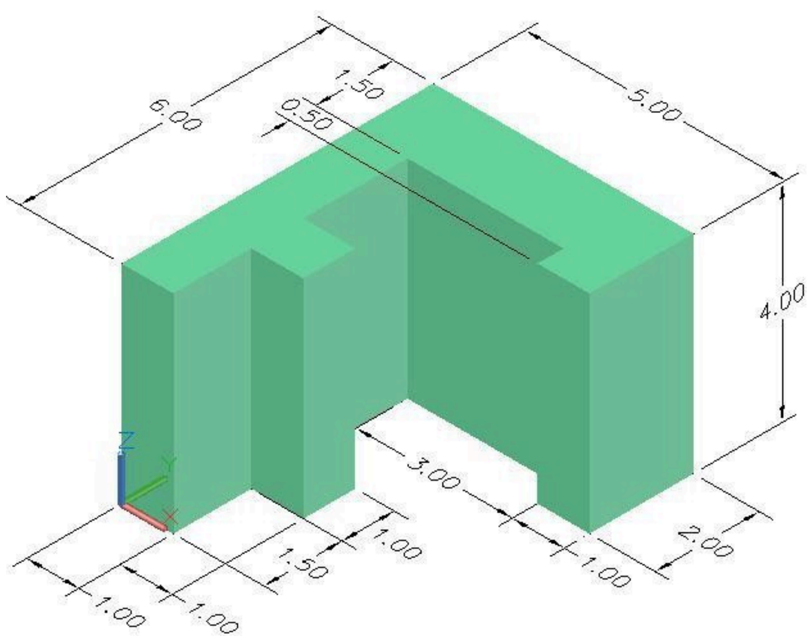


Figure Step 3B
3D Model [Click to see image full size]

Step 4

Insert the necessary driving dimensions to fully constrain the sketch. Add at least 1 driven dimension.

Step 5

Ensure that the sketch is fully constrained and all lines display purple. (Figure Step 5)

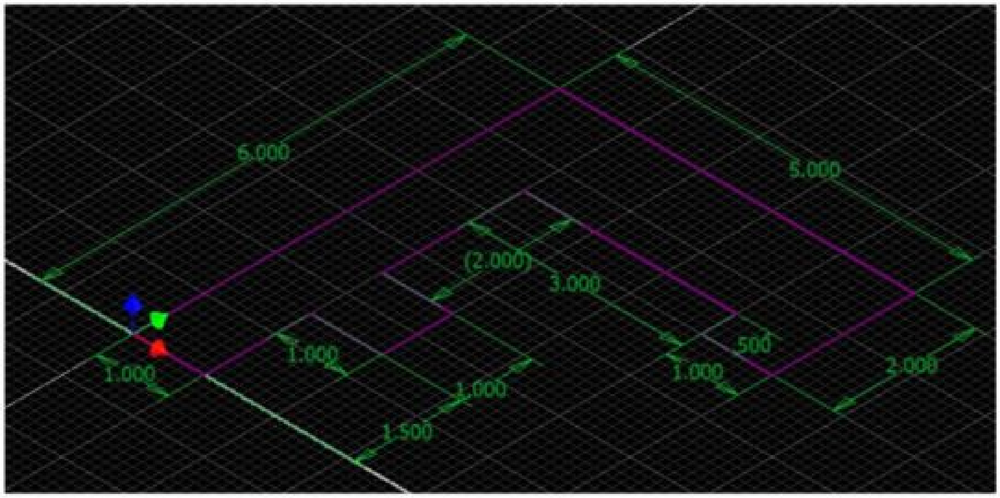


Figure Step 5
Fully Constrained Base Sketch [Click to see image full size]

Step 6

Extrude the sketch to complete the Base model and apply the color: Aluminum – Flat (Figure Step 6)

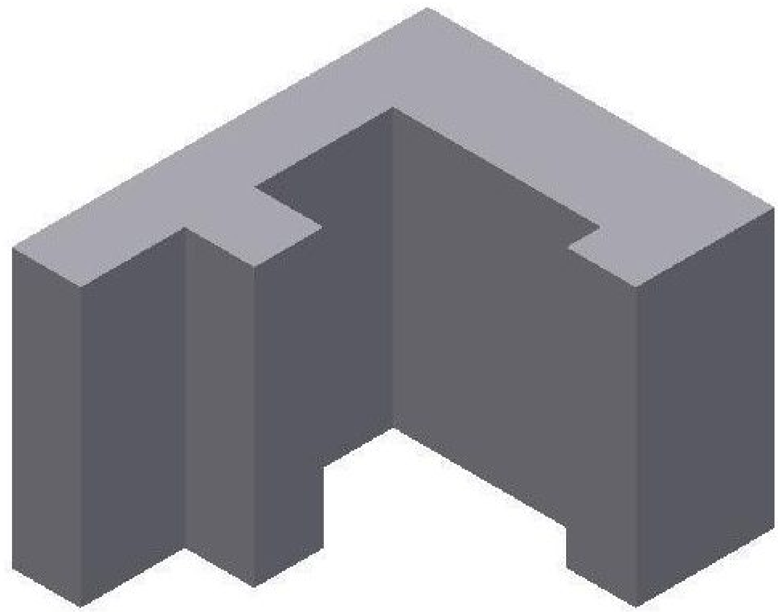


Figure Step 6
Completed Solid Model – Home View

Part 2

Module 7 Extruding – Part 2

Learning Outcomes

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

1. Describe how to construct a solid model using multiple sketches, linear dimensions, plus joining and cutting extrusions.
2. Describe and apply the 2DSKETCH command to create 2D sketches on a solid model. Extrude the sketches to either join them to or cut them from the solid model.

Point to Point Linear Dimensions

A *point to point linear* dimension is a dimension measuring the delta X or the delta Y distance between two points of an existing object or objects. The points, which are normally the endpoints of a line, could also be centre points of circles or arcs as you will see in future modules. If the points both lie on the same X or Y axis, the dimension can only inserted in one direction. If the points do not lie on the same axis, you have the choice of inserting either the delta X or the delta Y dimension. See Figure 7-1. The three steps used to insert a point to point linear dimension are shown below.

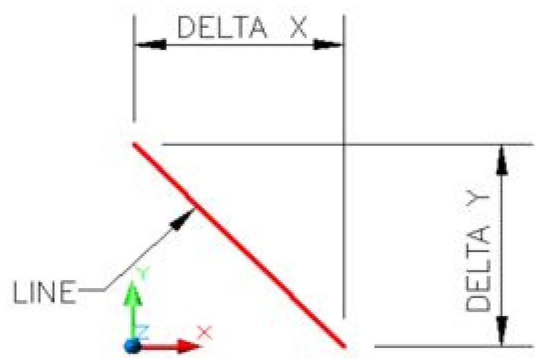
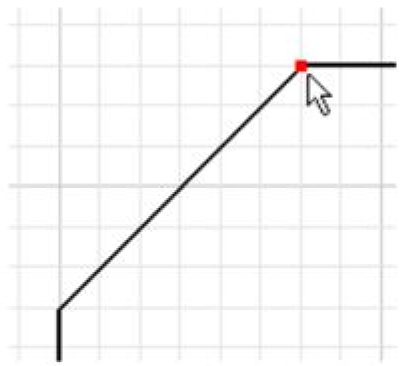
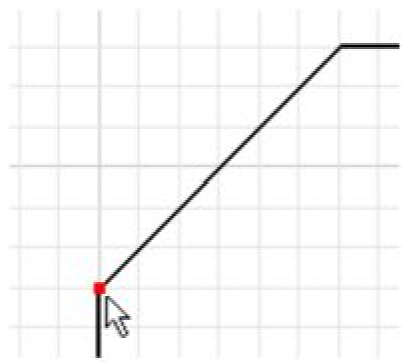


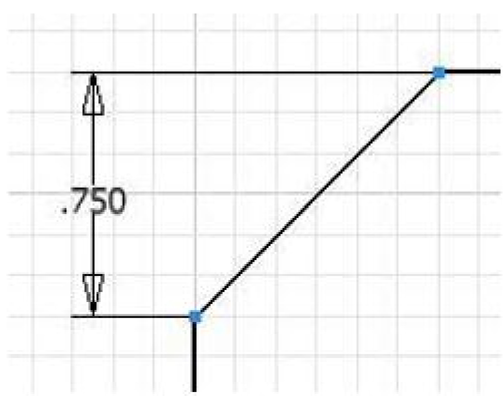
Figure 7-1
Linear Dimensions Applied To an Inclined Line



Step 1



Step 2

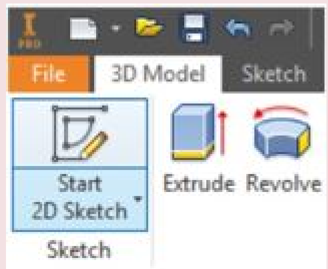


Step 3

Inventor Command: 2D SKETCH

The 2D SKETCH command is used to create a 2D sketch on a sketching plane or onto an existing 3D solid model.

Shortcut: **S**



WORK ALONG: Constructing a Solid Model Using Multiple Sketches

Step 1

Check the default project and if necessary, set it to Inventor Course.

Step 2

Enter the NEW command to start a new part file using the template: Metric-Modules Part (mm).ipt.

Step 3

Save the file with the name: Inventor Workalong 07-1. (Figure Step 3A and 3B)

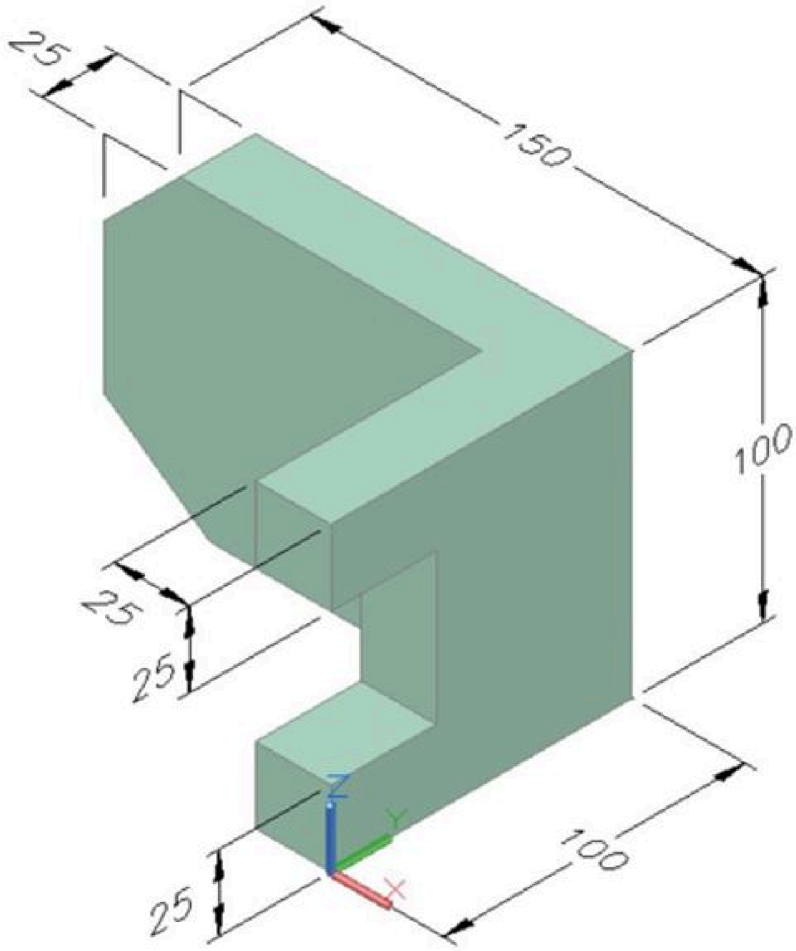


Figure Step 3A
Solid Model – Home View

AUTHOR'S COMMENTS: Two views of the model that you will be constructing in this workalong are shown below. Note the location of X0Y0Z0.

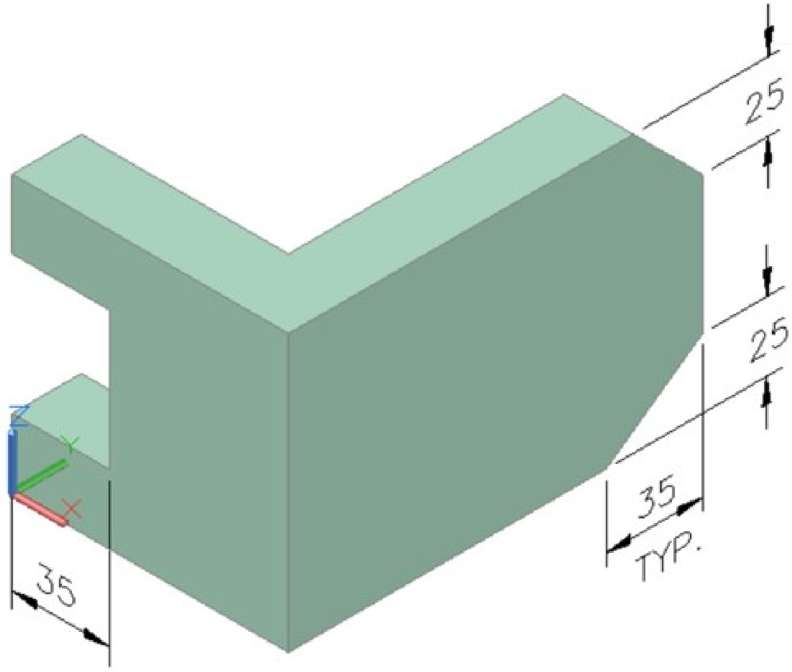


Figure Step 3B
Solid Model – Rotated View

Step 4

Edit Sketch 1 and enter the PROJECT GEOMETRY command and project the Center Point onto the sketching plane. Press Esc to exit the command. (Figure Step 4)

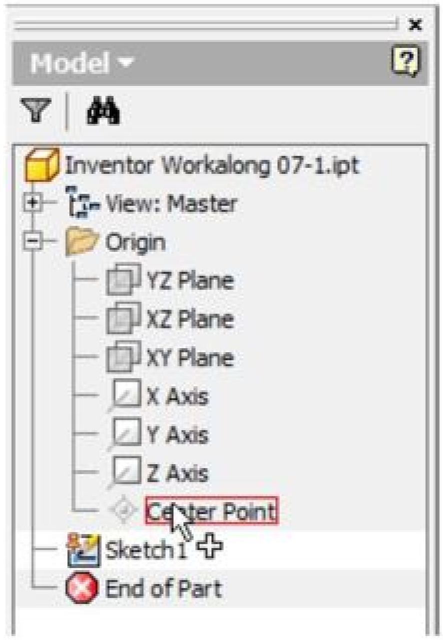


Figure Step 4

Step 5

Draw the Top view of the model starting at X0Y0Z0. This is the Base sketch. (Figure Step 5)

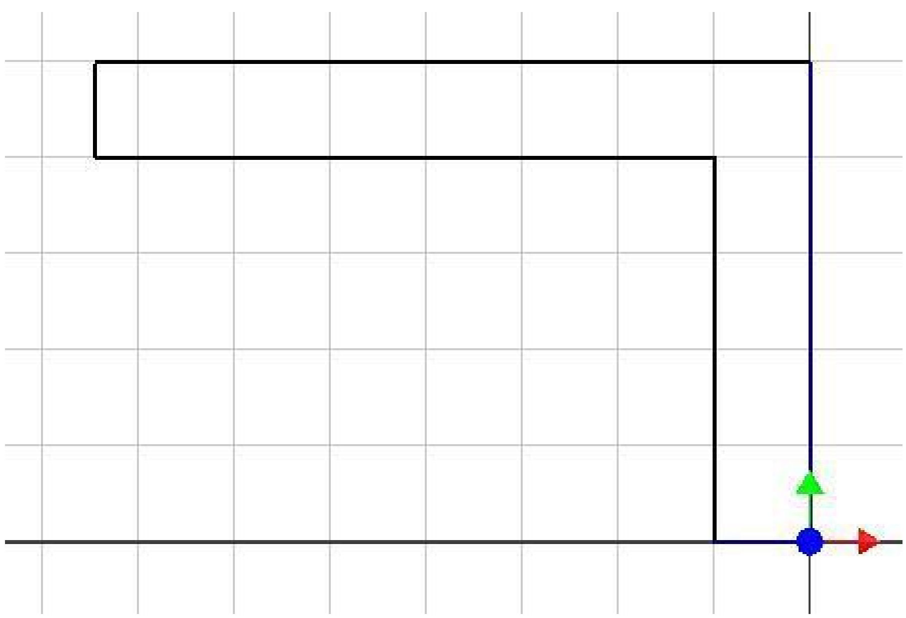


Figure Step 5

AUTHOR'S COMMENTS: Since the most contoured shape of the model is the Top view, this is the view the should be drawn first.

Step 6

Add 4 driving dimensions to fully constrain the sketch. Add 2 driven dimensions.

Step 7

Press F8 to display the constraints. (Figure Step 7)

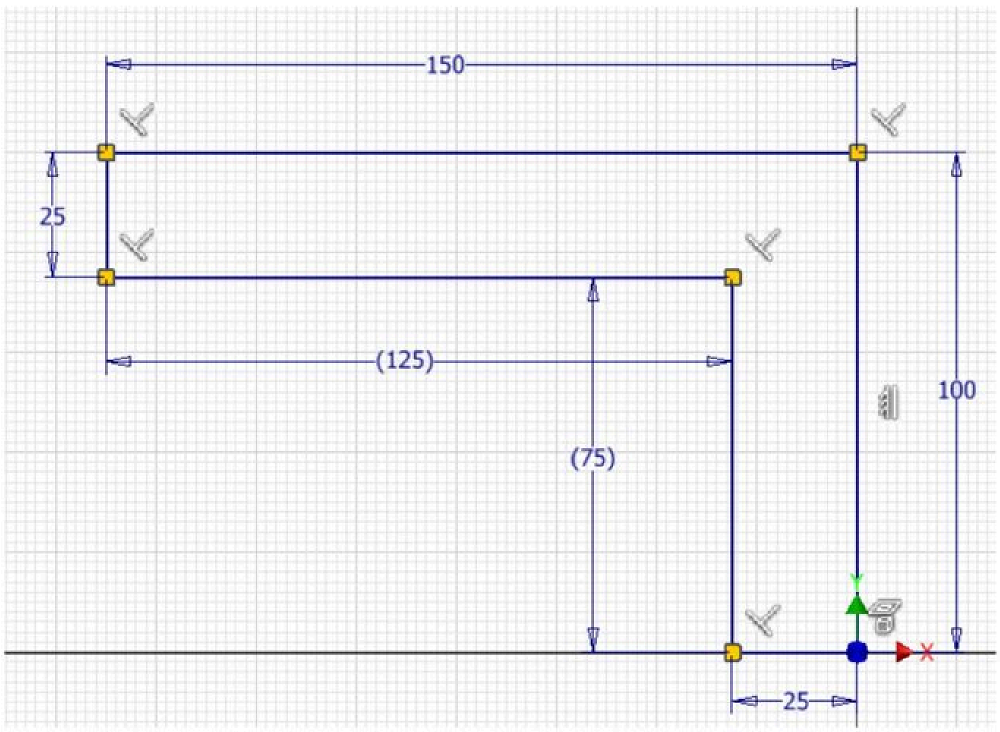


Figure Step 7

Step 8

The sketch should now be fully constrained and all the lines display purple on a black background. (Figure Step 8)

Figure Step 8

Step 9

Press F6 to change the view to the Home view. Click the FINISH SKETCH command to return to Model mode. (Figure Step 9)

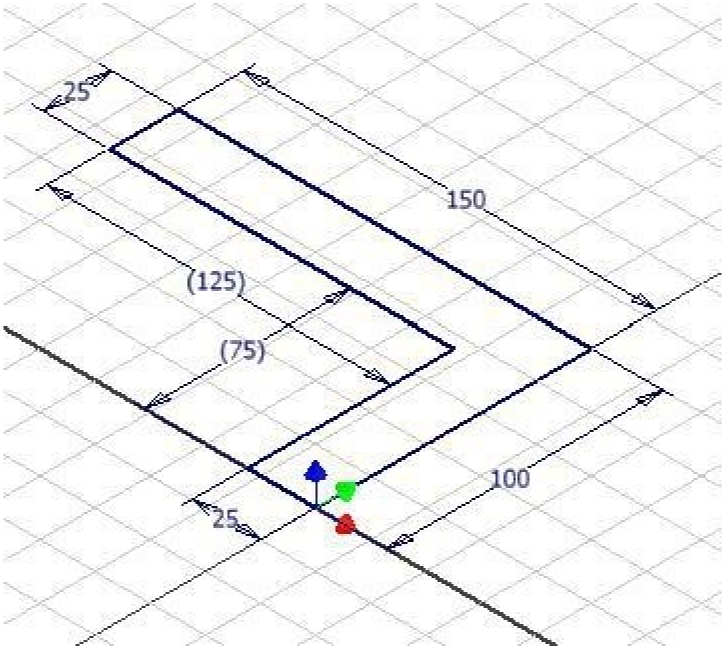


Figure Step 9

Step 10

Enter E to execute the EXTRUDE command and extrude the model 100 mm in the positive Z direction. (Figure Step 10)

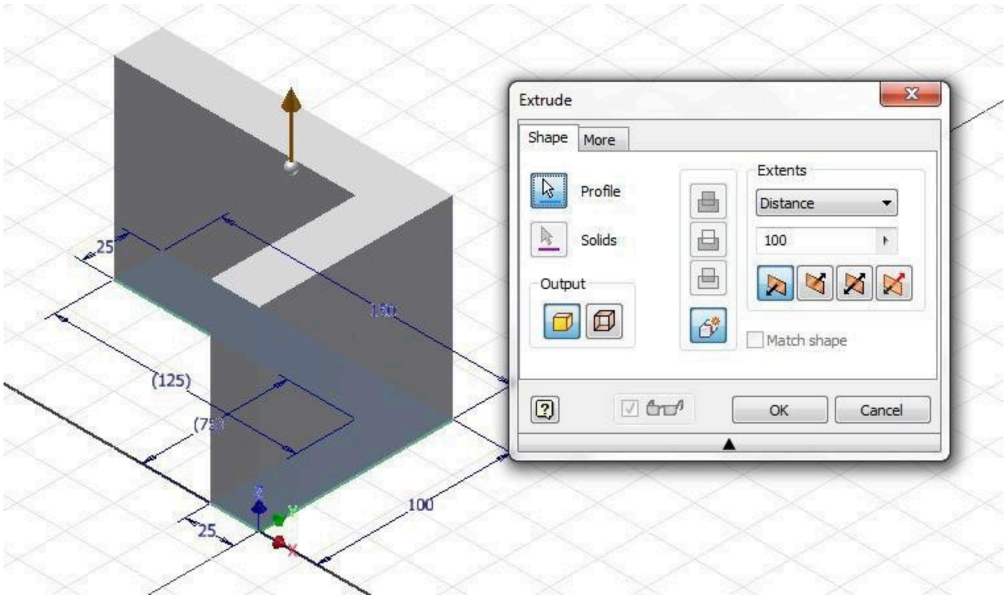


Figure Step 10 [Click to see image full size]

Step 11

The solid model should appear as shown in the figure. (Figure Step 11)

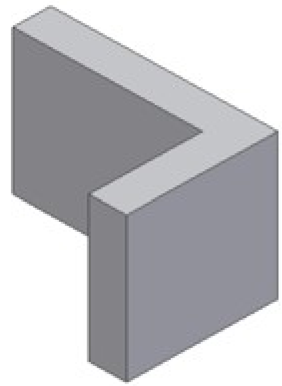


Figure Step 11

Step 12

Click the 2D SKETCH icon and when prompted, select the right side plane as shown in the figure. Exit the command and the grid will display on the plane in Sketch mode. Press F6 to change to the Home view. (Figure Step 12A, 12B, and 12C)

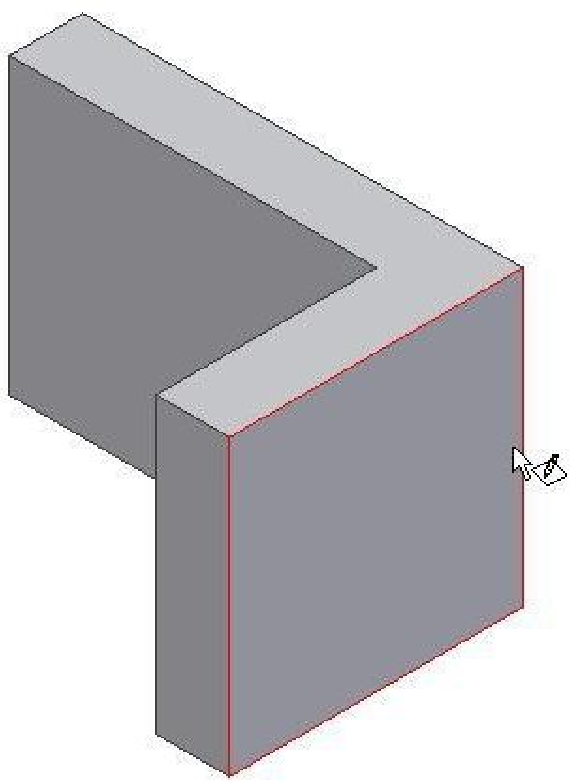


Figure Step 12A

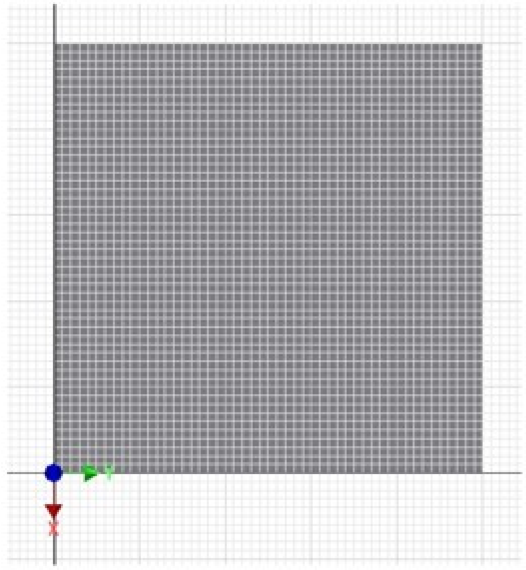


Figure Step 12B

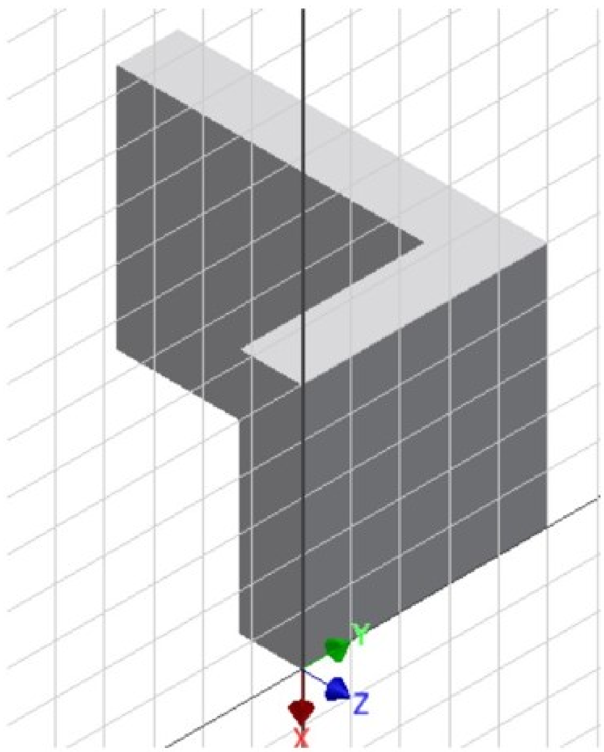


Figure Step 12C

Step 13

Enter L for the LINE command and start the first endpoint of the line by snapping onto the plane edge line. (Figure Step 13)

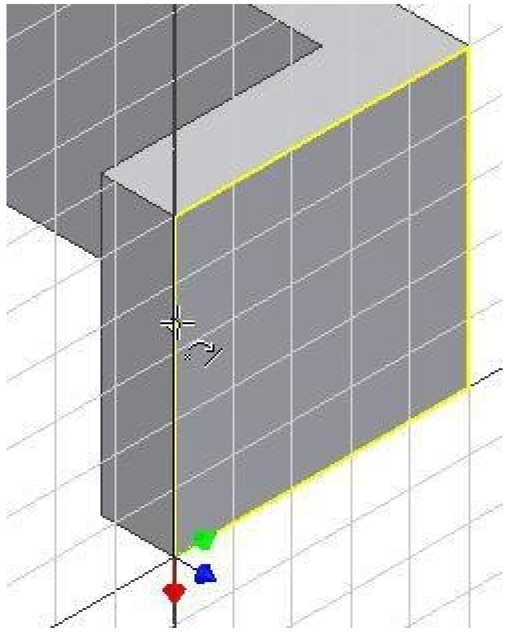


Figure Step 13

AUTHOR'S COMMENTS: The exact location on the line is not important. Guess at a location about 25 millimeters from the top. It MUST, however, be snapped onto the edge.

Step 14

Draw three lines, applying perpendicular geometrical constraints and snapping back onto the edge. Guess at the length of the lines making them approximately the correct length. (Figure Step 14)

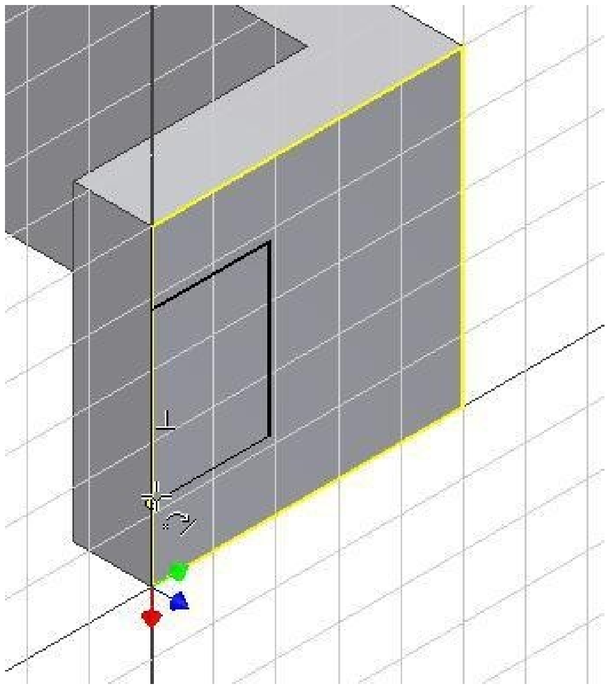


Figure Step 14

Step 15

Press F8 to display the constraint icons. (Figure Step 15)

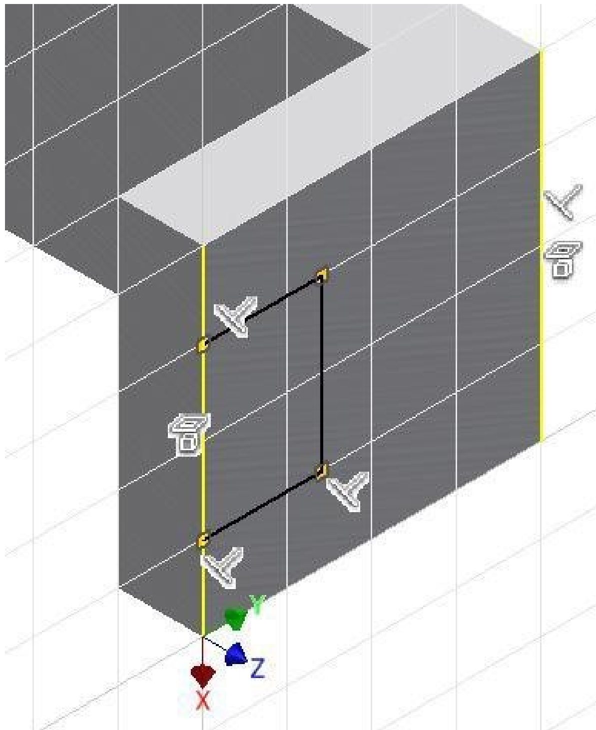


Figure Step 15

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

Step 16

Enter D for the GENERAL DIMENSION command. Move the cursor to bottom corner of the model. When a small point highlights, click the mouse. (Figure Step 16)

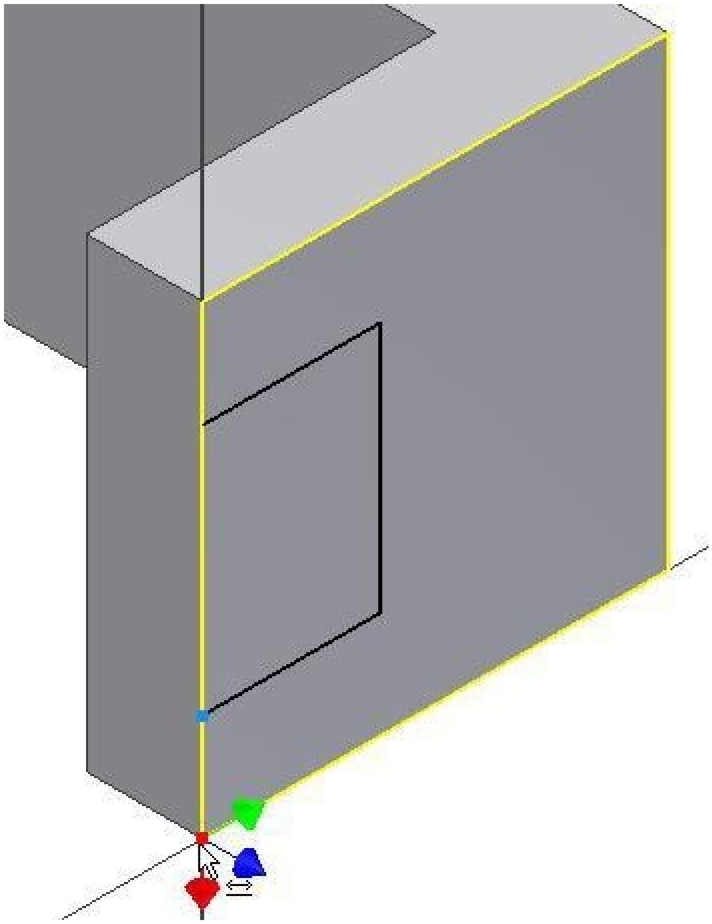


Figure Step 16

Step 17

For the second point of the dimension, click the end of the line as shown in the figure. Locate and set the dimension to 25. (Figure Step 17A and 17B)

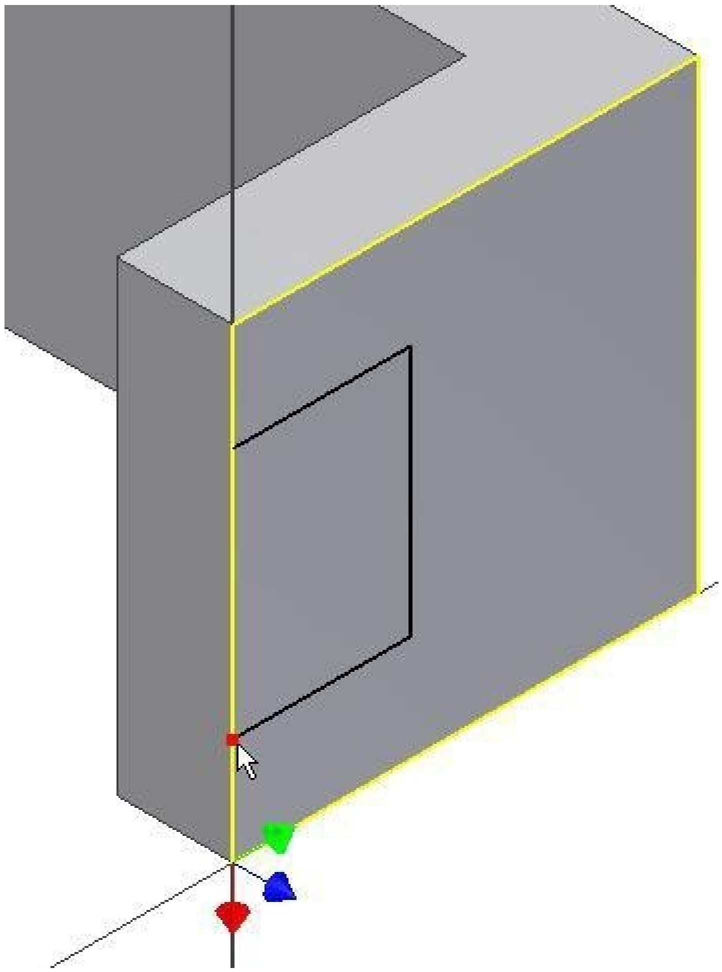


Figure Step 17A

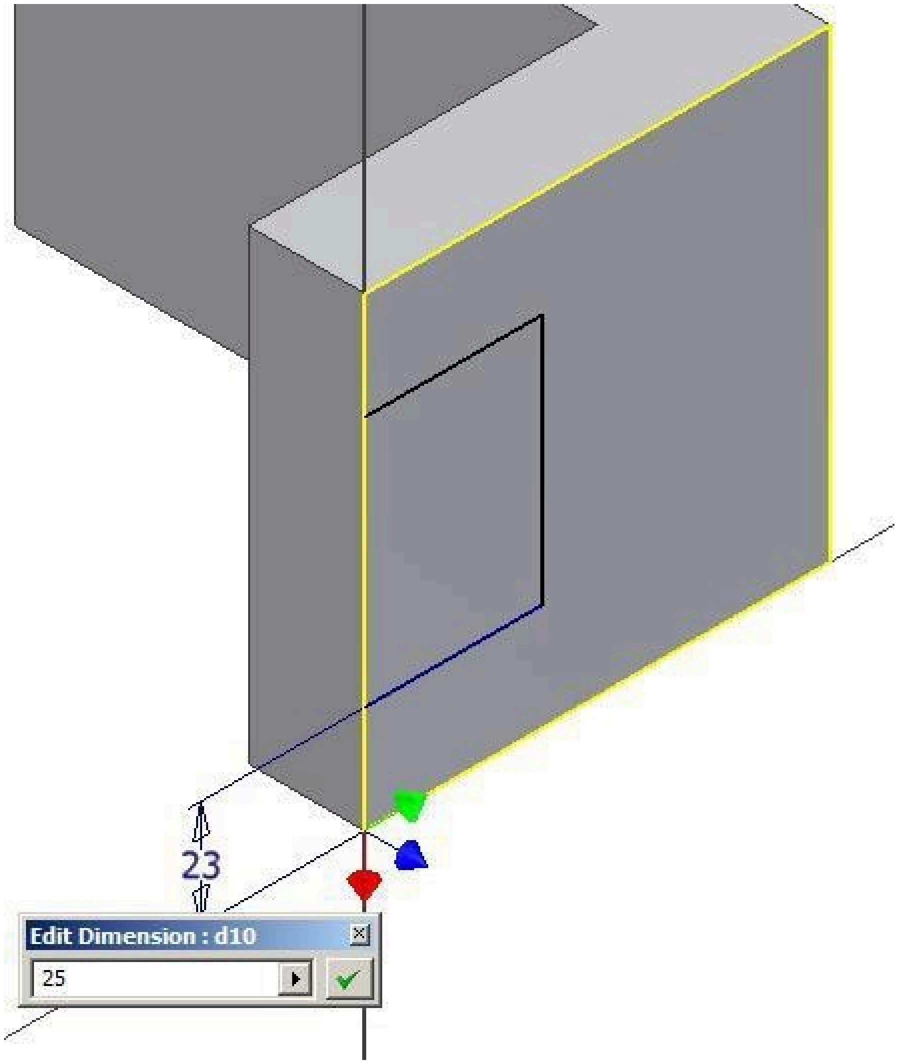


Figure Step 17B

Step 18

Add two additional dimensions to fully constrain it. (Figure Step 18A, 18B, and 18C)

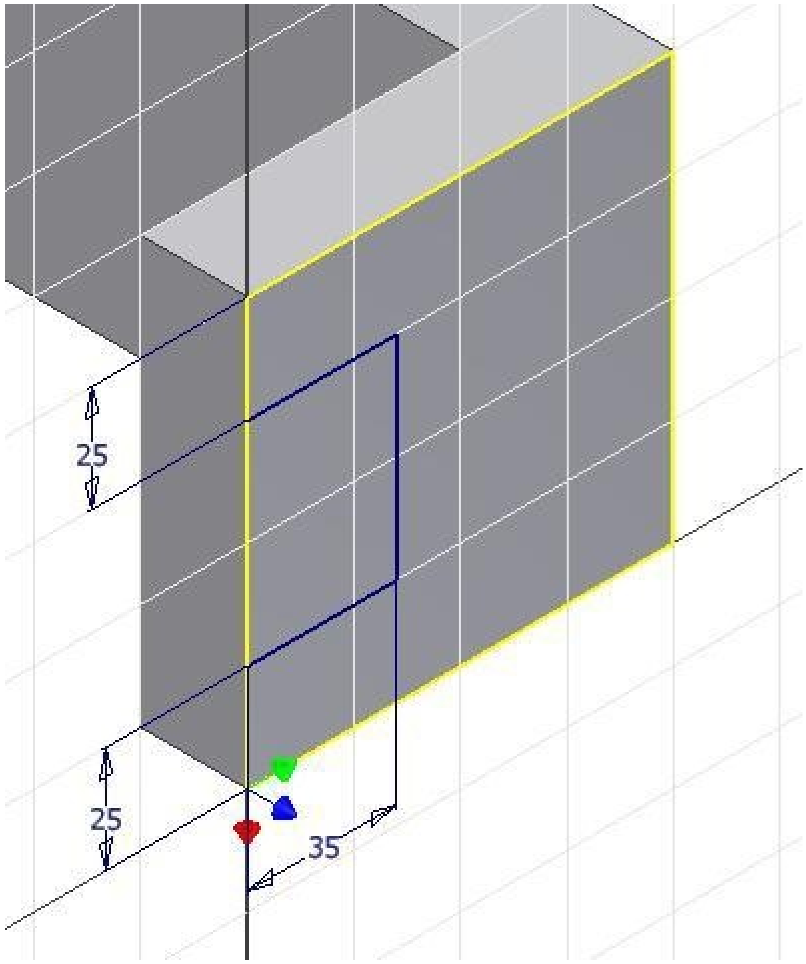


Figure Step 18A

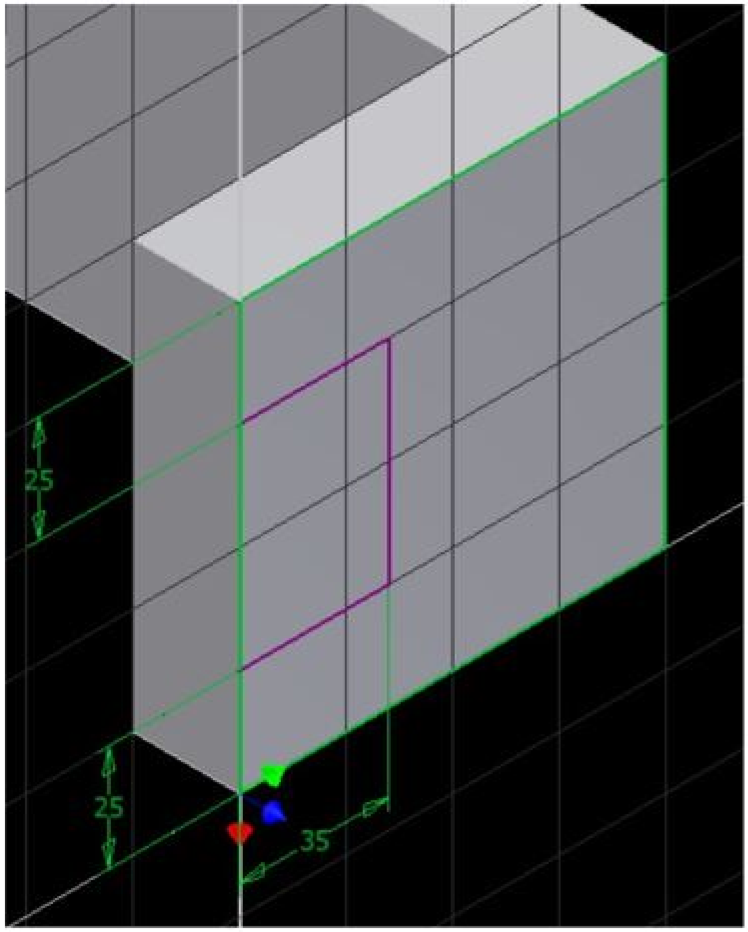


Figure Step 18B

Figure Step 18C

Step 19

Enter E for the EXTRUDE command. (Figure Step 19)

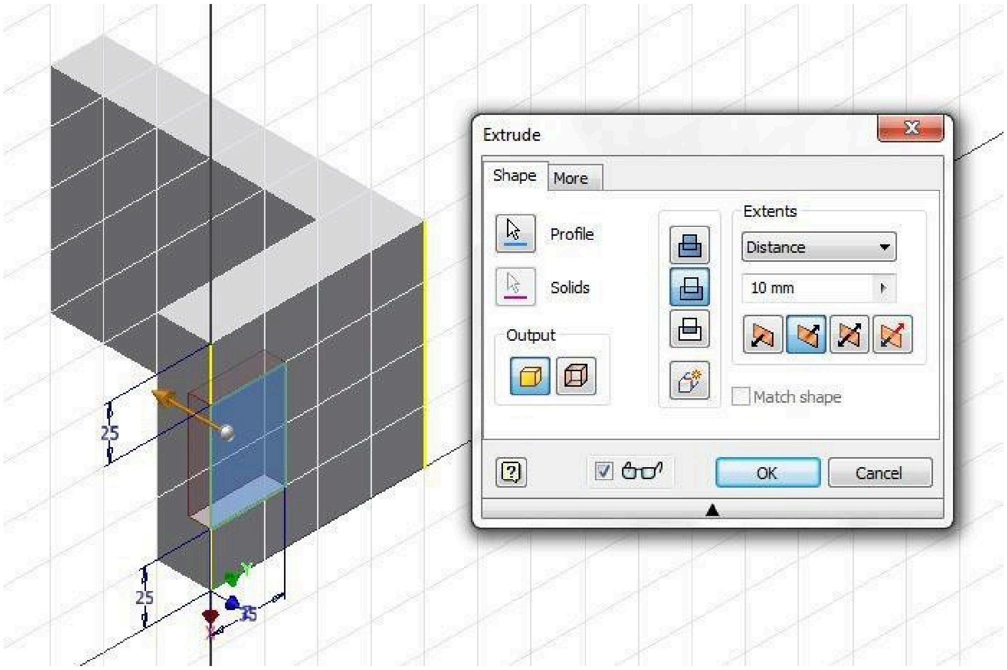


Figure Step 19 [Click to see image full size]

Step 20

Set the Extents to To, the type to Cut and select the back face to locate where to extrude to. (Figure Step 20)

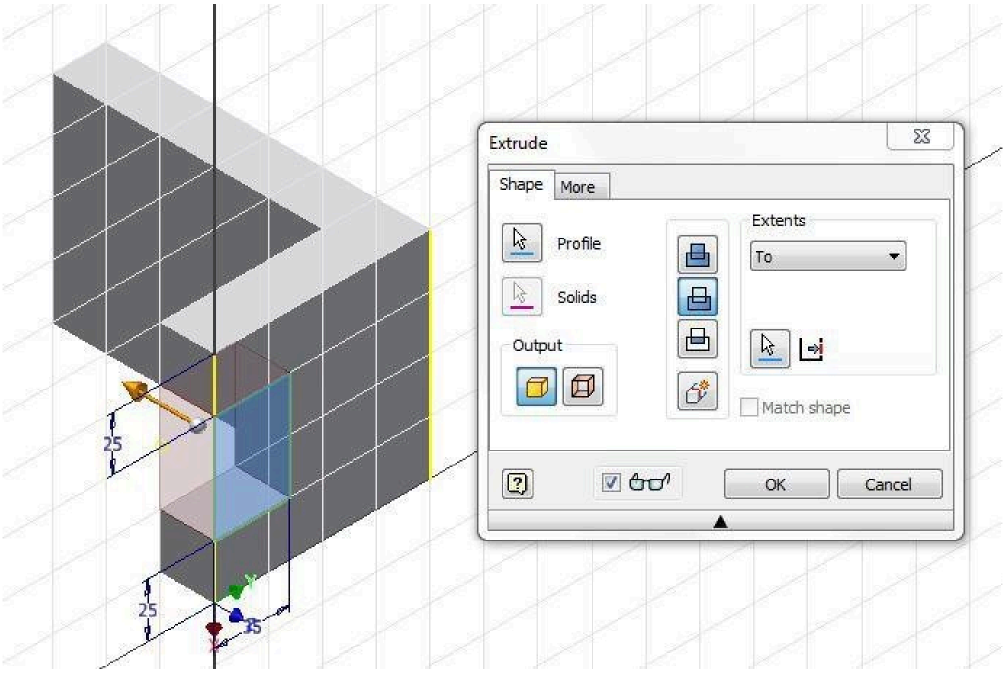


Figure Step 20 [Click to see image full size]

Step 21

Your model should appear as shown in the figure. (Figure Step 21)

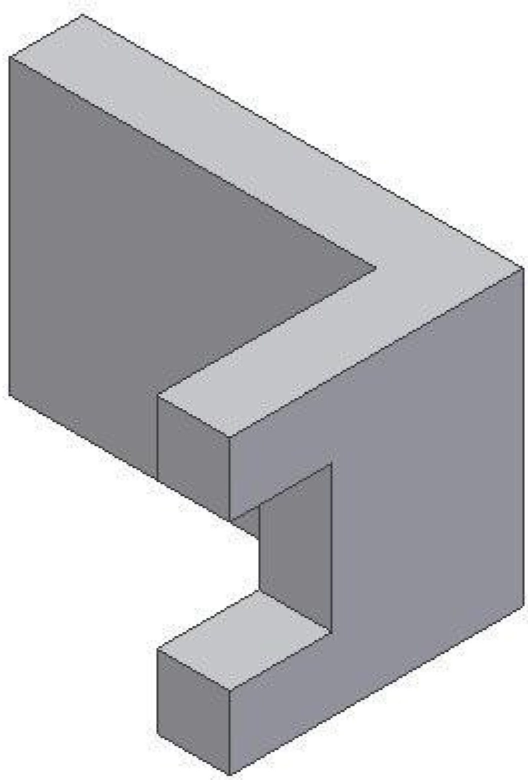


Figure Step 21

Step 22

Click the 2D Sketch icon and select the top plane to start a new sketch. Orbit the model to match the figure. (Figure Step 22)

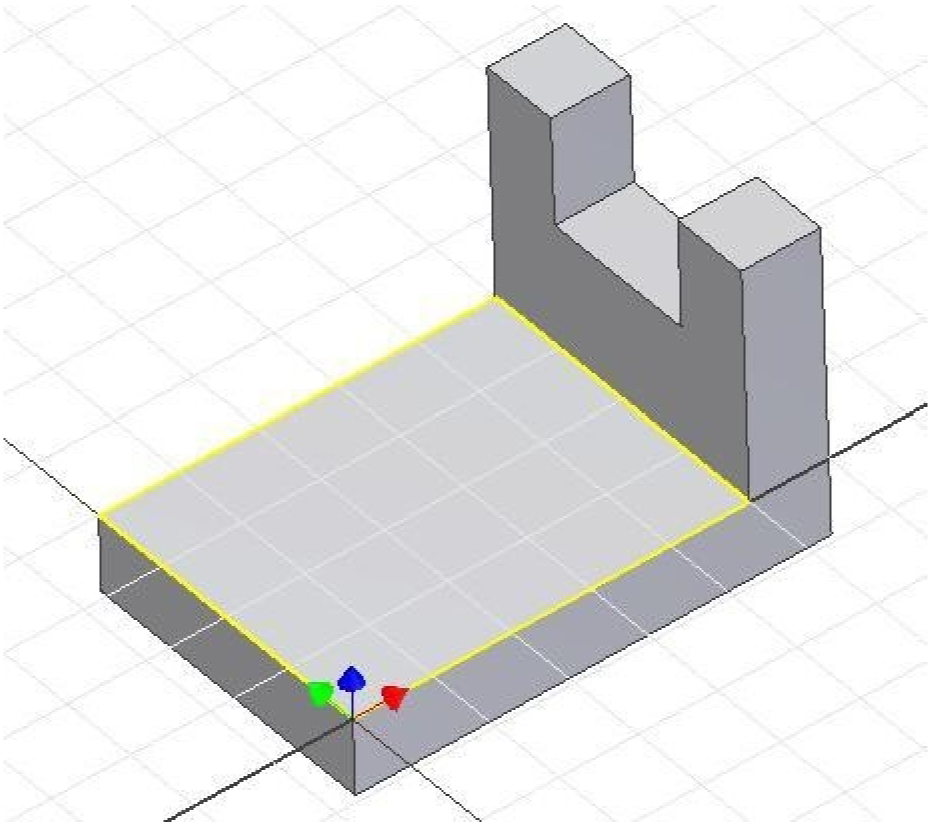


Figure Step 22

Step 23

On the sketch, draw a line across the corner. Make sure that you snap both ends of the line to the edges. Guess at the distance from the corner. (Figure Step 23)

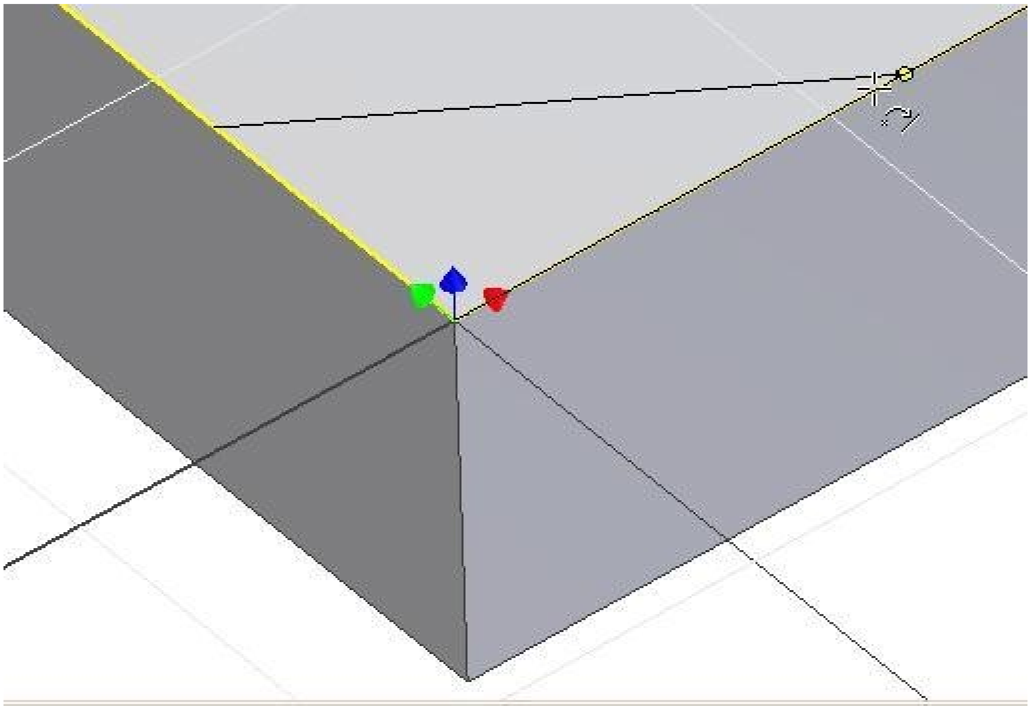


Figure Step 23

Step 24

Draw a line on the opposite side and dimension both lines using four point to point dimensions. (Figure Step 24)

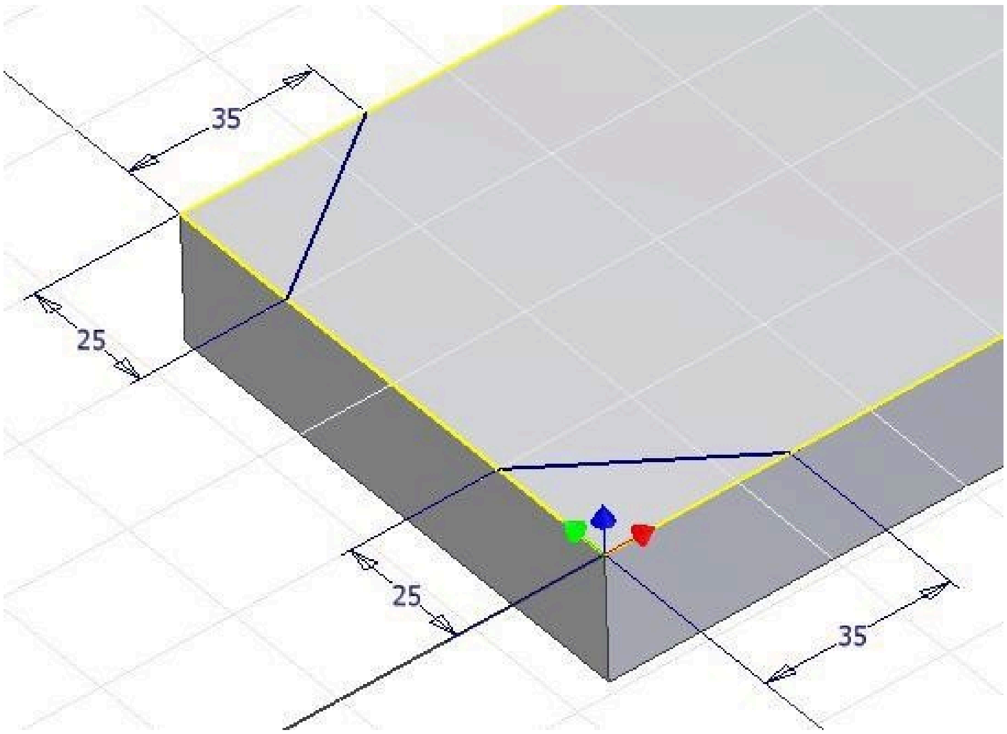


Figure Step 24 [Click to see image full size]

Step 25

Your sketch should be fully constrained as shown in the figure. (Figure Step 25)

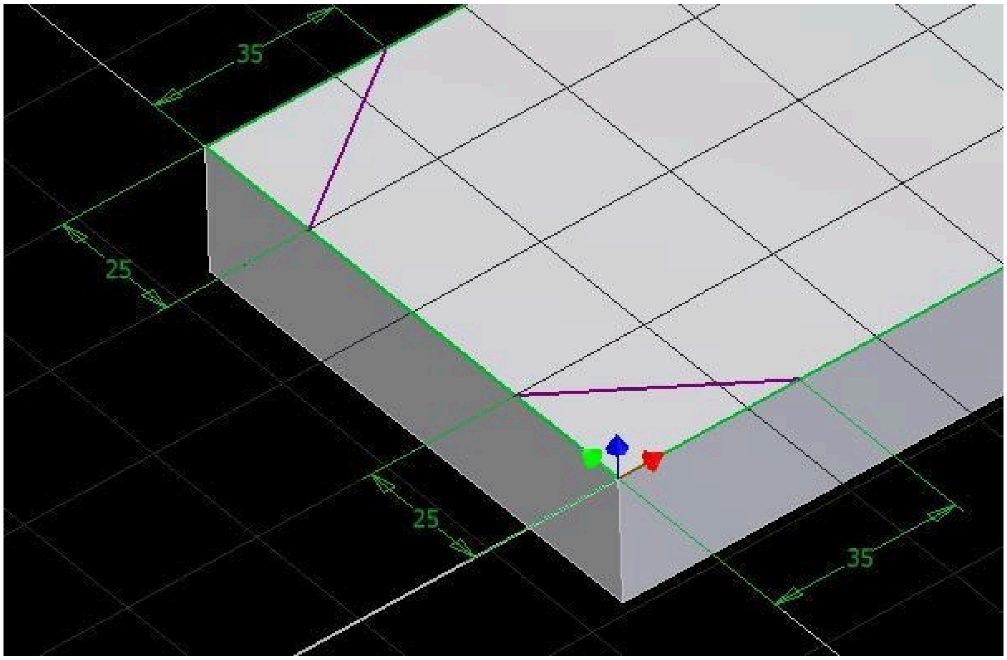


Figure Step 25 [Click to see image full size]

Step 26

Press F8 to display the constraint icons. (Figure Step 26)

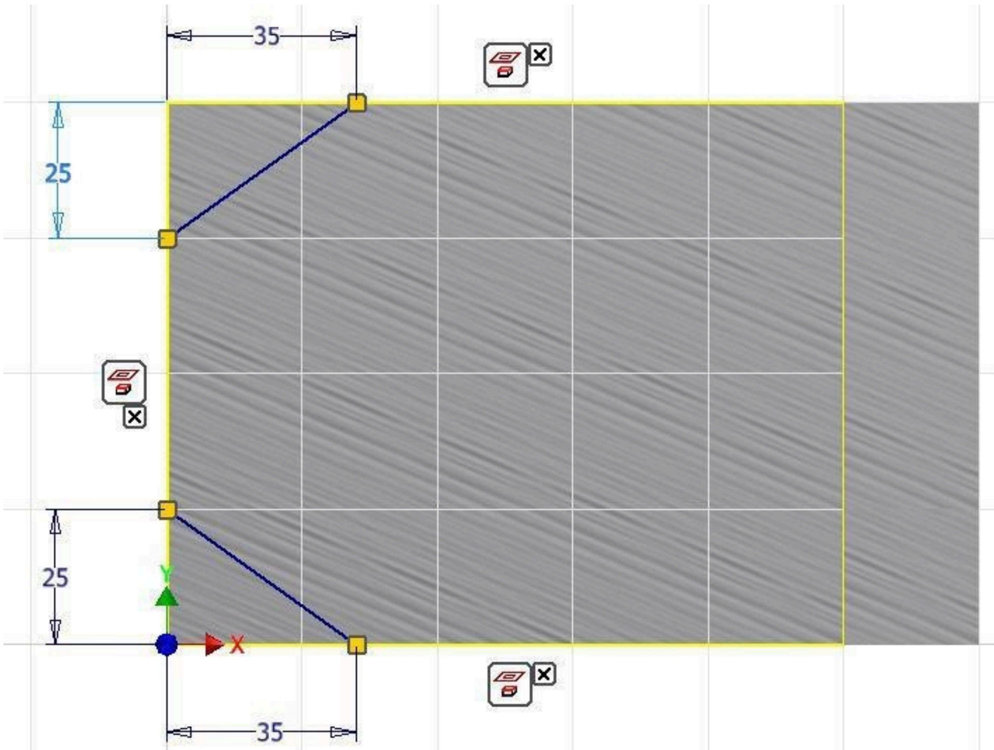


Figure Step 26 [Click to see image full size]

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

Step 27

Extrude the sketch using the To Next extents. Select the two profiles to extrude and ensure that you enable the Cut icon. (Figure Step 27)

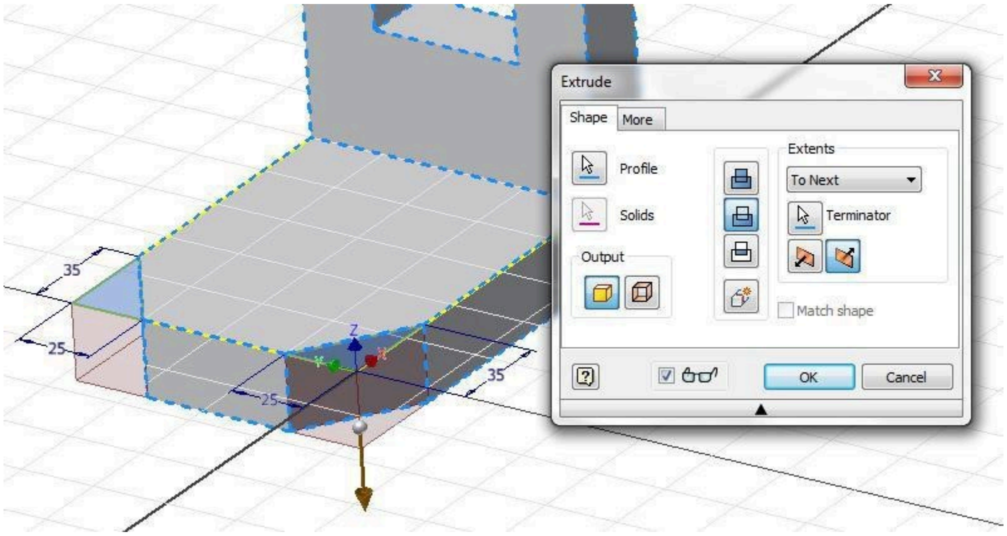


Figure Step 27 [Click to see image full size]

USER TIP: Drawing lines on the plane of a 3D model works a little different then drawing lines on a 2D sketch before the model is extruded. In most cases, Inventor will only allow you to draw one line segment at time when you are drawing a line on the 3D model. Watch the prompts in the Status bar and you will know when you have to reenter the start point again or you can draw the next segment continuous.

Step 28

Change to the Home view by pressing F6. Change the colour of the completed model to color: Light Steel Blue. (Figure Step 28)

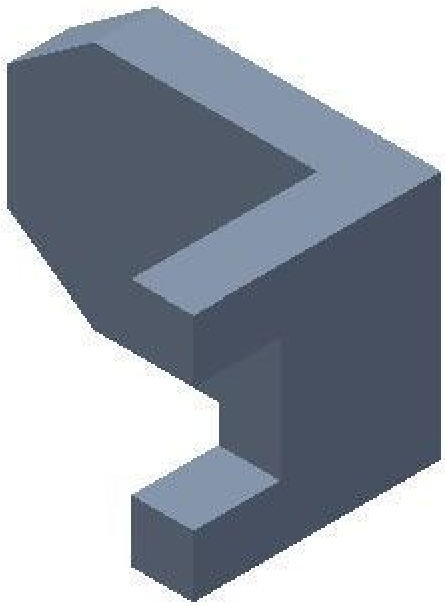


Figure Step 28

Step 29

Expand the Browser bar. You can see that the model hierarchy shows the three sketches and an extrusion of each one. (Figure Step 29)

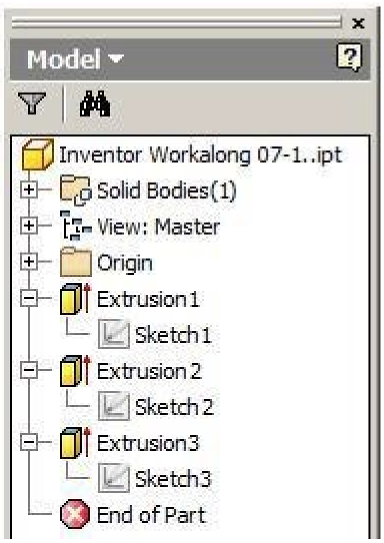


Figure Step 29

Step 30

Save and close the file.

Key Principles

Key Principles in Module 7

1. A point to point linear dimension is a dimension measuring the delta X or the delta Y distance between two points of an existing object or objects.
2. The 2D SKETCH command is used to create a 2D sketch on a sketching plane or onto an existing 3D solid model.Lab Exercise 7.1

Lab Exercise 7-1

Time allowed: 45 minutes.

Part Name	**Project**	**Units**	**Template**	**Color**	**Material**
Inventor Lab 07-1	Inventor Course	Millimeters	Metric-Modules Part (mm).ipt	Steel	N/A

Step 1

Project the Center Point onto the sketching plane.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude them to produced the solid model. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. (Figure Step 2A and 2B)

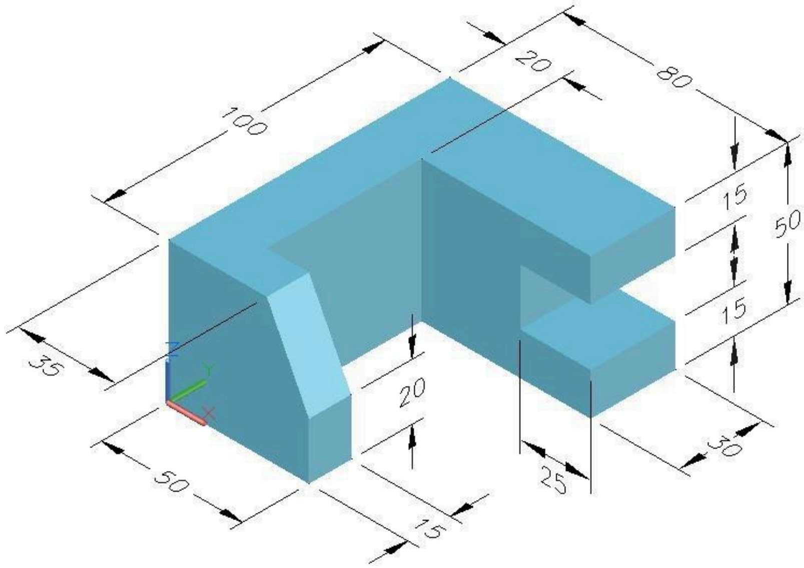


Figure Step 2A
3D Model

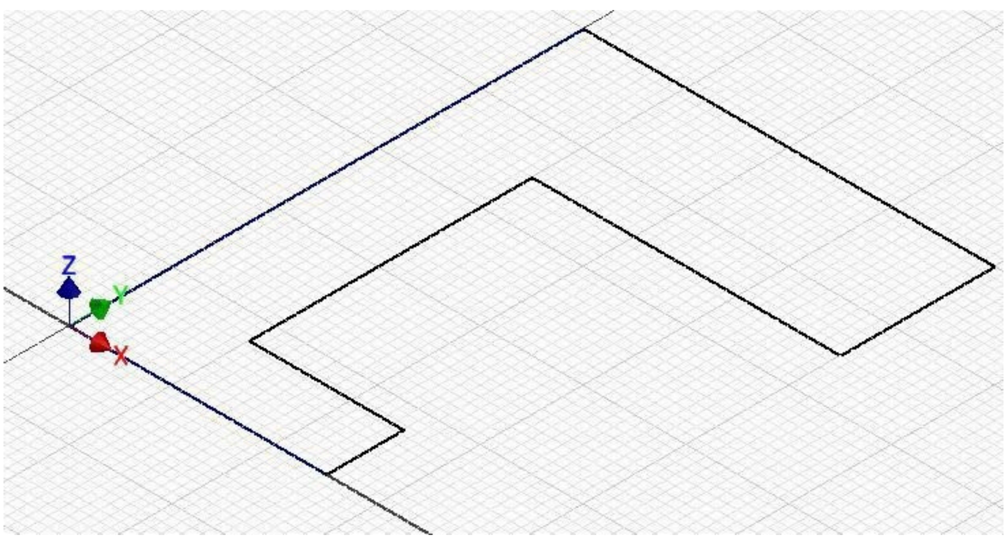


Figure Step 2B
Suggested Base Sketch

Step 3

Apply the colour shown above. (Figure Step 3)

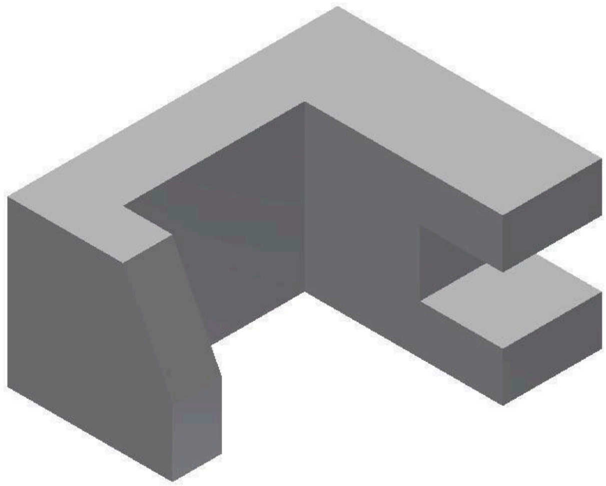
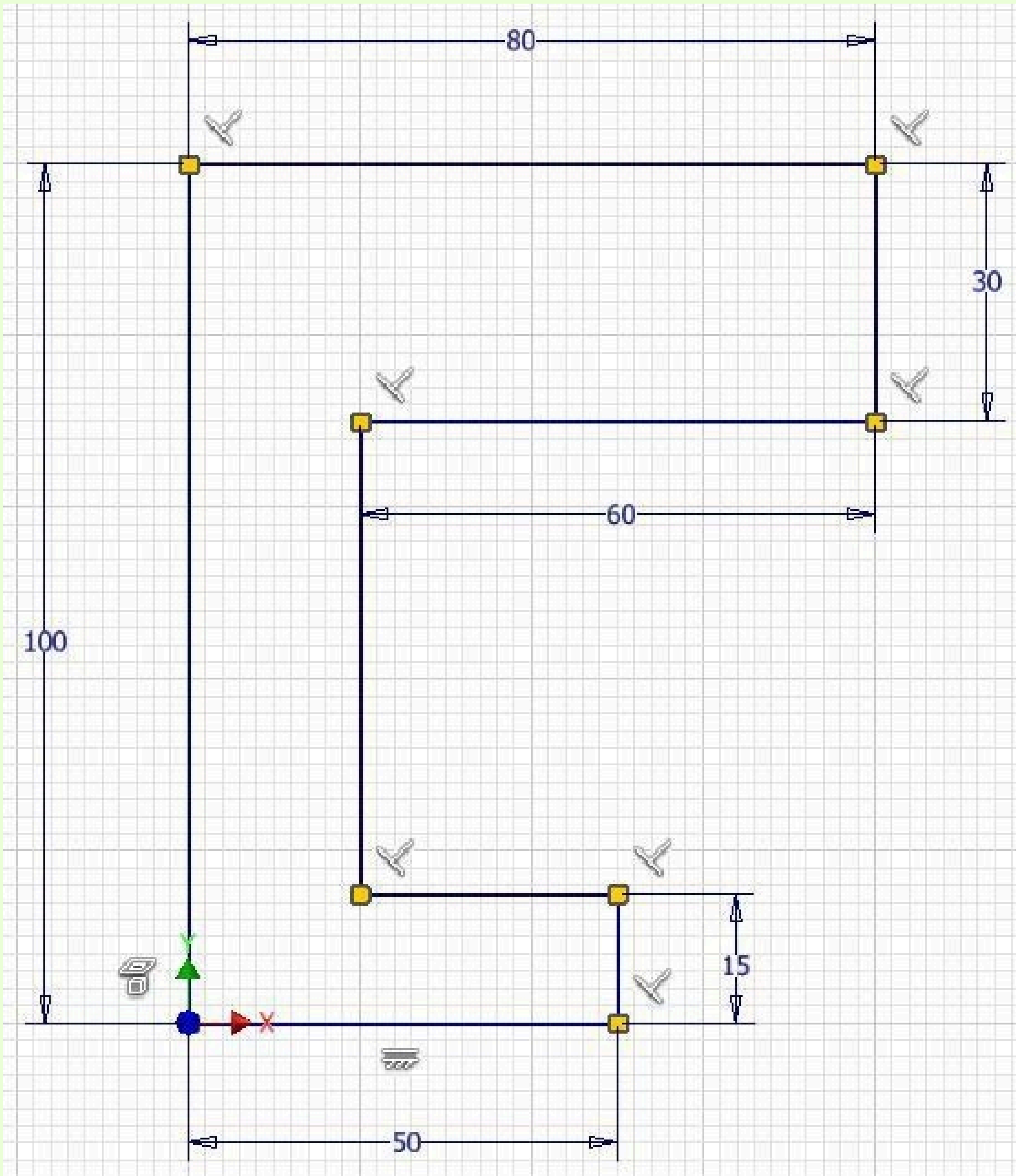
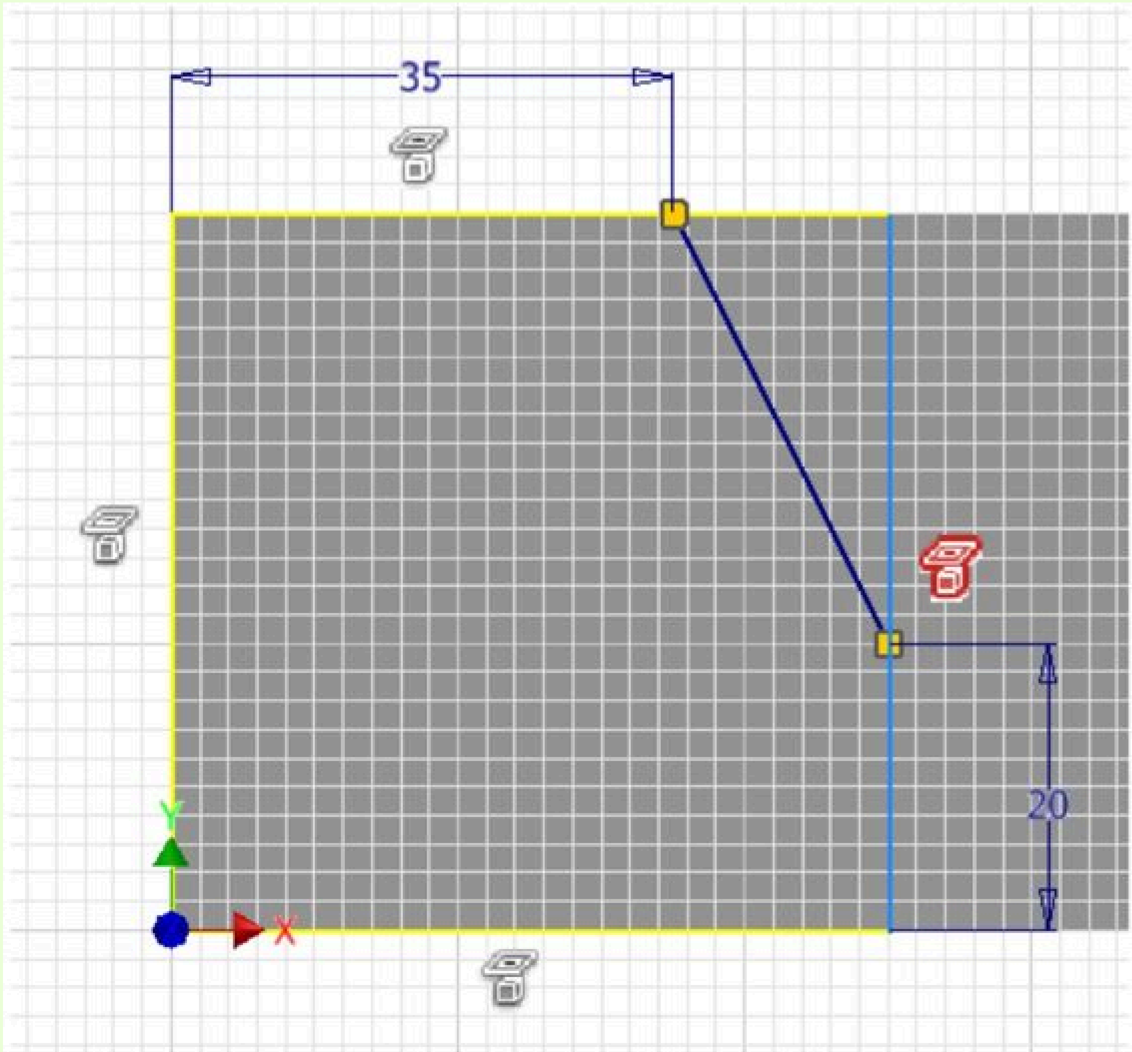


Figure Step 3
Completed Solid Model – Home View

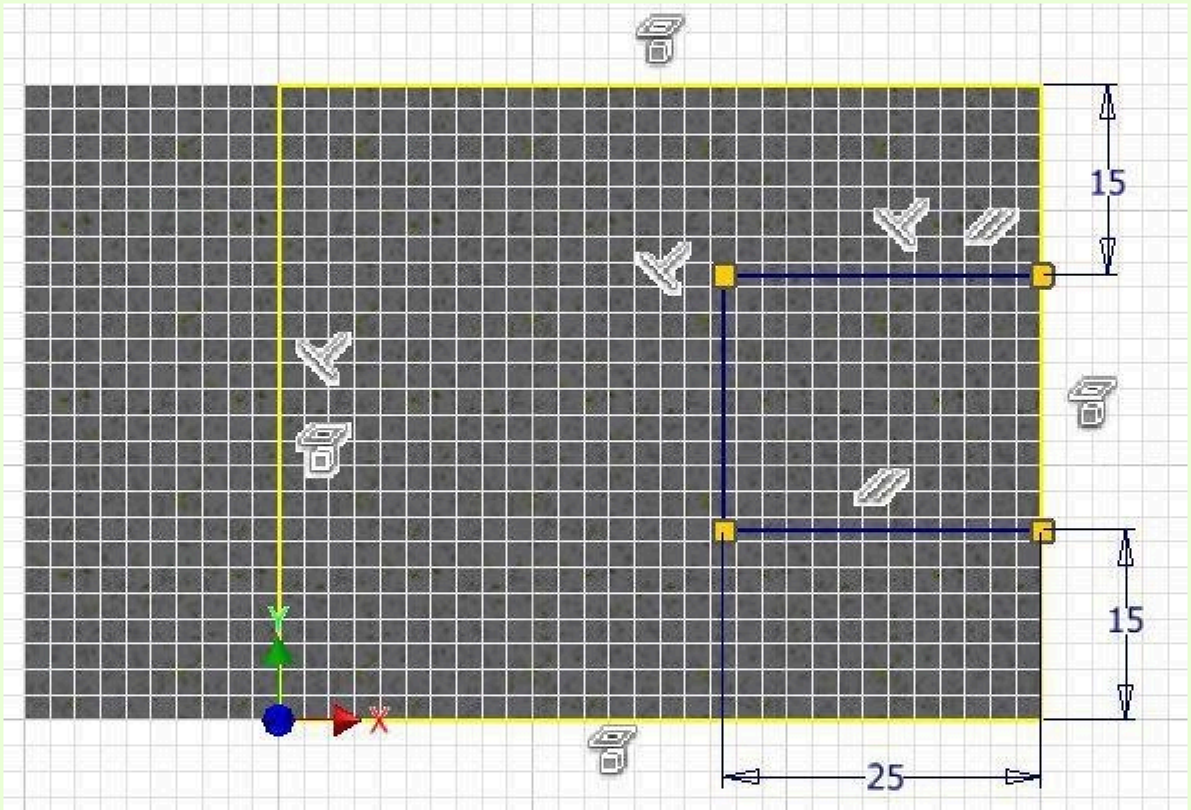
AUTHOR'S GEOMETRIC CONSTRAINS: The following three figures shows the base and additional sketch's construction method plus geometric and dimensional constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketches using different construction methods and constraints, that is what is important. You may want to compare your construction method and constraints used with the authors.



Author's Base Sketch



Author's Sketch



Author's Sketch

Lab Exercise 7-2

Time allowed: 45 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab 07-2	Inventor Course	Millimeters	Metric-Modules Part (mm).ipt	Chrome – Polished	N/A

Step 1

Project the Center Point onto the base sketching plane.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude them to produced the solid model. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. (Figure Step 2A, 2B, and 2C)

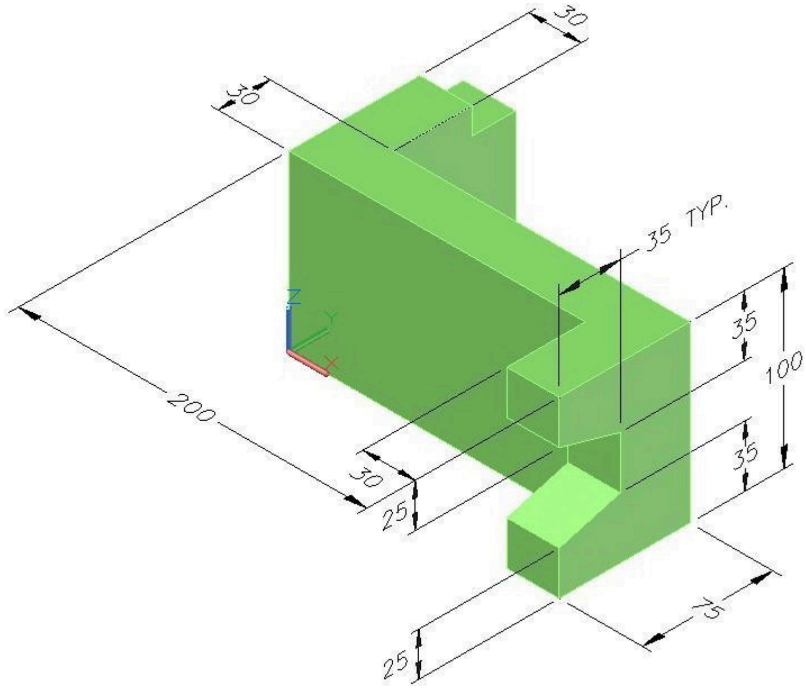


Figure Step 2A
3D Model – Home View

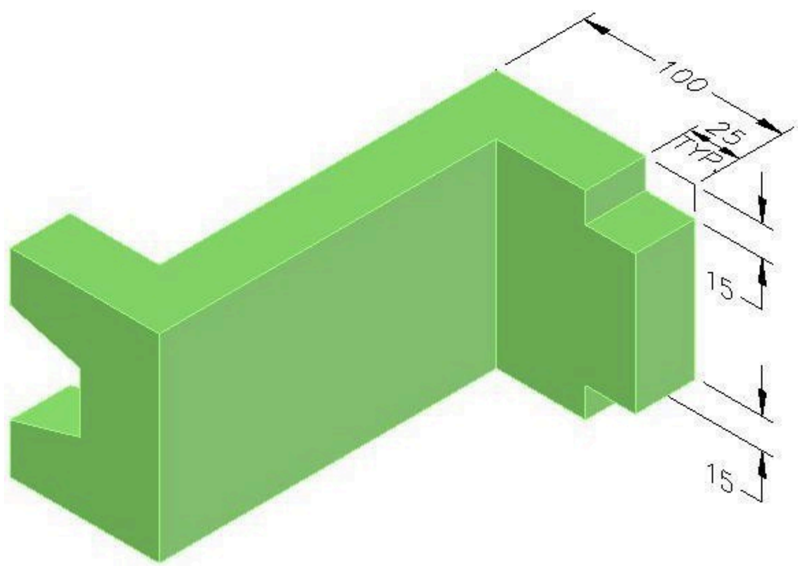


Figure Step 2B
A Rotated View of the 3D Model

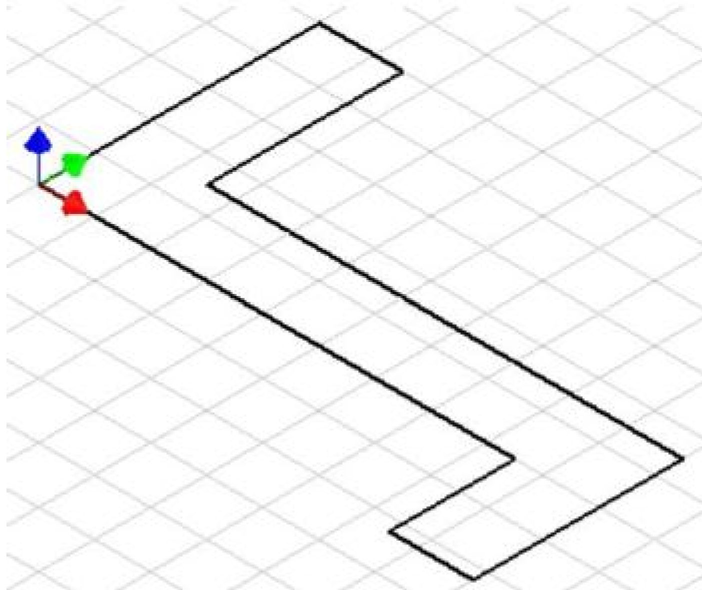


Figure Step 2C
Suggested Base View

Step 3

Apply the colour shown above. (Figure Step 3)

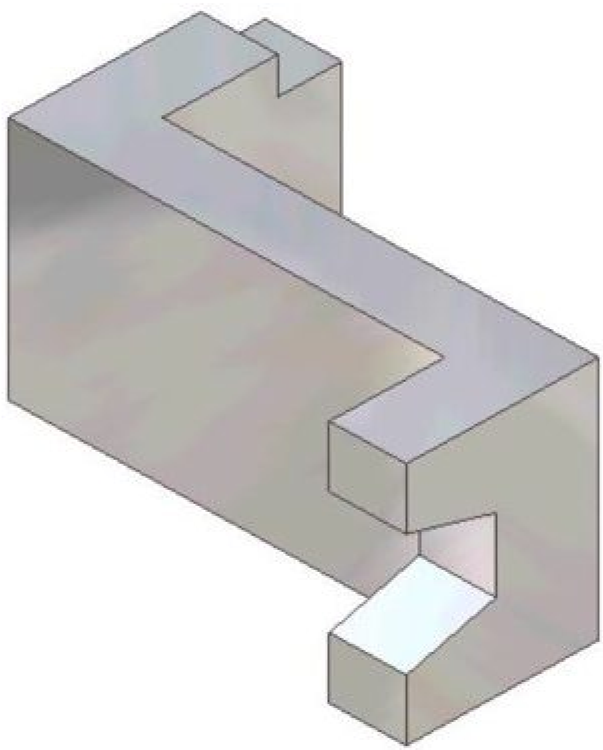
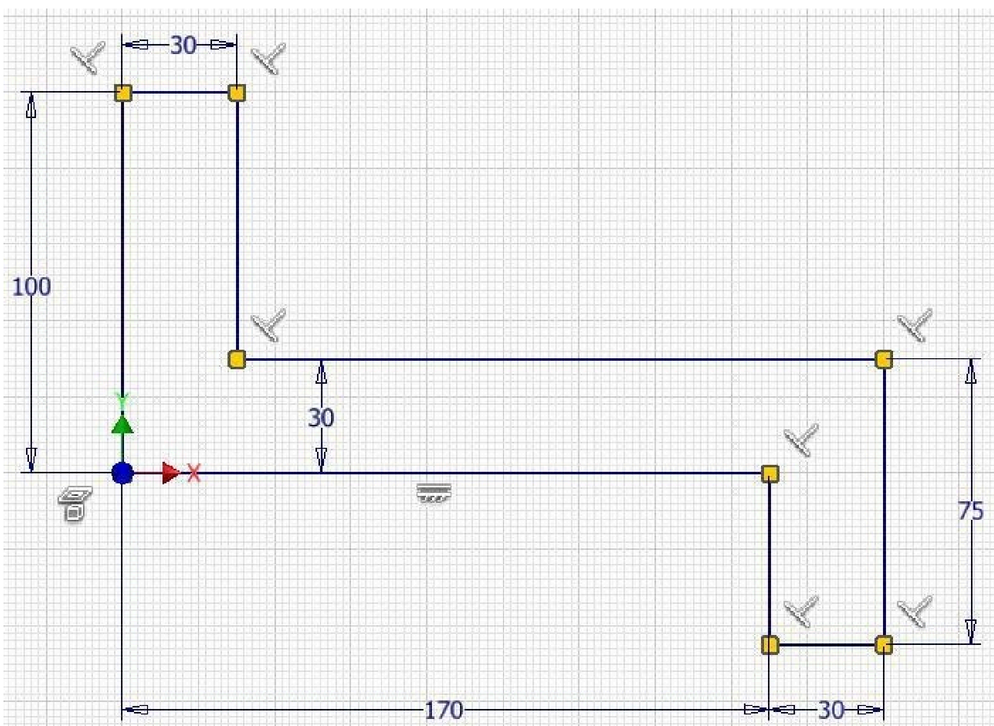


Figure Step 3
Completed Solid Model – Home View

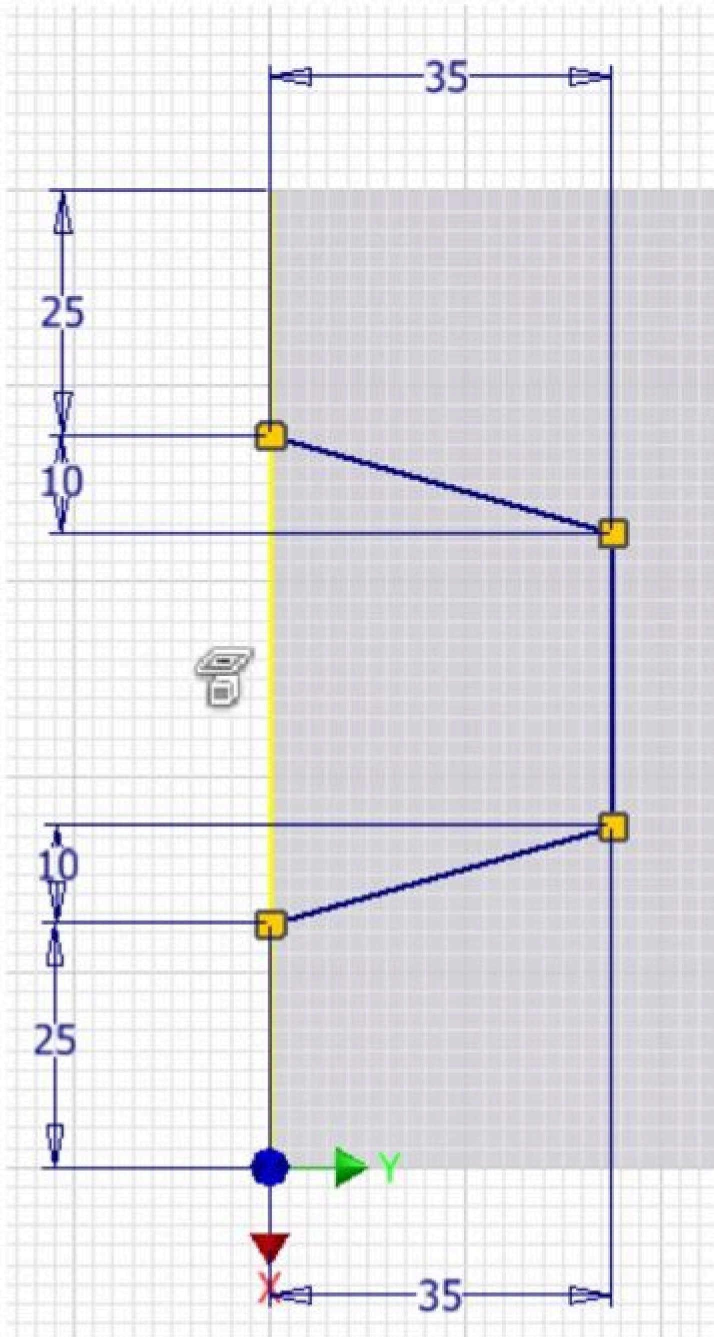
Step 4

Save the file with the name: Inventor Lab 07-2 as shown above.

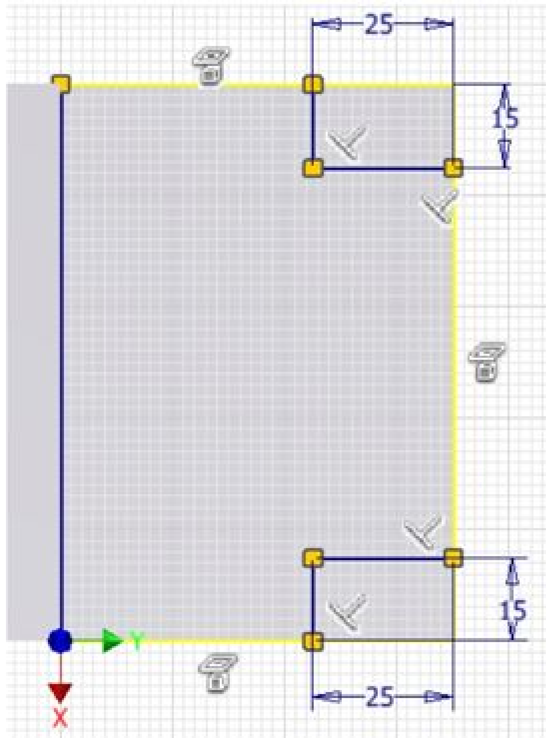
AUTHOR'S GEOMETRIC CONSTRAINTS: The following three figures shows the base and additional sketch's construction method plus geometric and dimensional constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketches using different construction methods and constraints, that is what is important. You may want to compare your construction method and constraints used with the authors.



Author's Base Sketch [Click to see image full size]



Author's Sketch



Author's Sketch

Module 8 Multiview Drawings

Learning Outcomes

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

1. Describe multiview drawings, the glass box principle, the three standard views, object lines, and hidden lines.
2. From a 3D pictorial of an object, draw a multiview drawing using the three standard views.

NOTE: If you understand multiview drawings, object lines, hidden lines, and you can draw the three standard views of an object, skip this module.

Multiview Drawing

The drafting and design world uses a system of representing a three- dimensional object by drawing two-dimensional views. It is called a *multiview drawing.* To explain this system of drawing, the object shown in Figure 8-1 will be used in this module. To draw a two-dimensional view of one side of the object, place a imaginary plane parallel to the side and project the view of the object perpendicular onto the plane. This is called *orthographic projection*. Imagine the plane to a sheet of glass. See Figure 8-2.

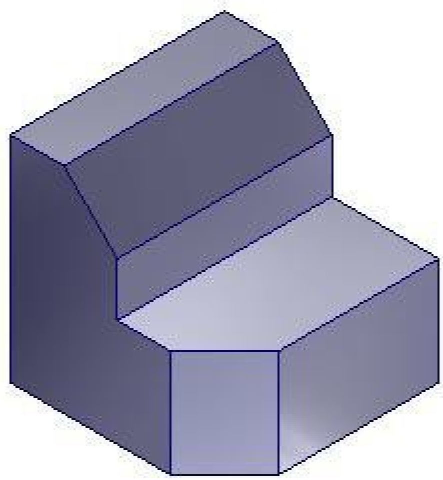


Figure 8-1
The 3D Model

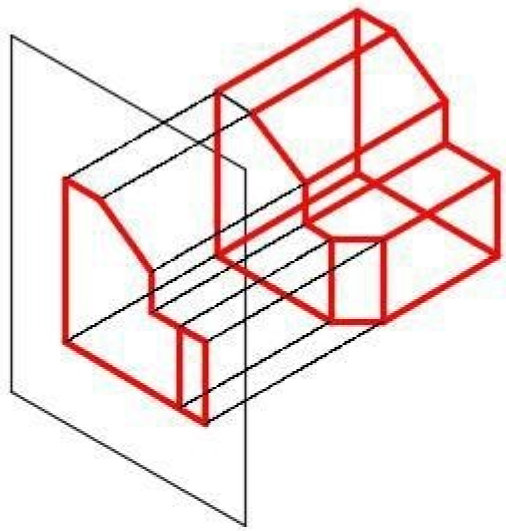


Figure 8-2
Projecting the
Two-Dimensional View

The Glass Box Principle

To carry this principal further, place a plane on each side of the object for a total of six planes or sheets of glass to form a glass box. This is called the *Glass Box Principle*, see Figure 8-3. Picture unfolding the glass box onto a flat two-dimensional plane as shown in Figure 8-4. All six views are now visible at the same time.

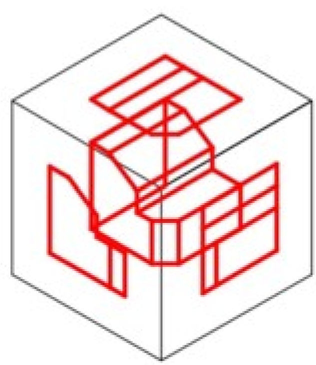


Figure 8-3
The Glass Box Principle

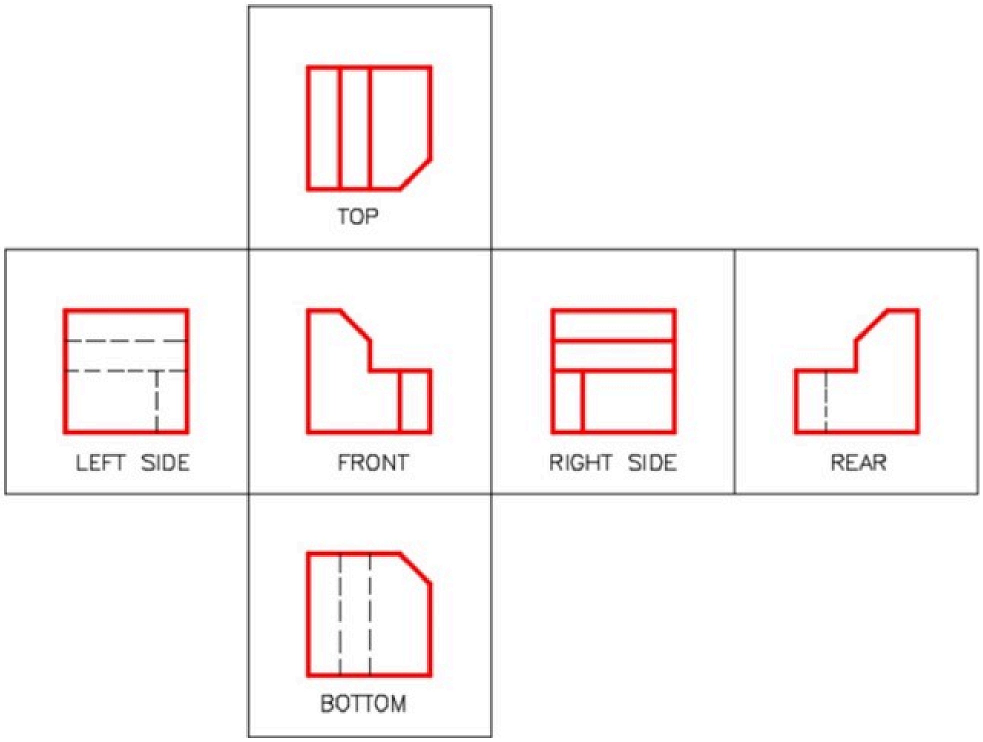


Figure 8-4
The Glass Box Unfolded [Click to see image full size]

The Three Standard Views

In almost all objects, three views are adequate to describe it. In fact, there are many objects that only need two views and some that only need one view to describe it. The six views are Top, Front, Right Side, Left Side, Rear, and Bottom. The three standard views are the *Top*, *Front*, and *Right Side*. They must be drawn in the positions shown in Figure 8-5 and they must be aligned.

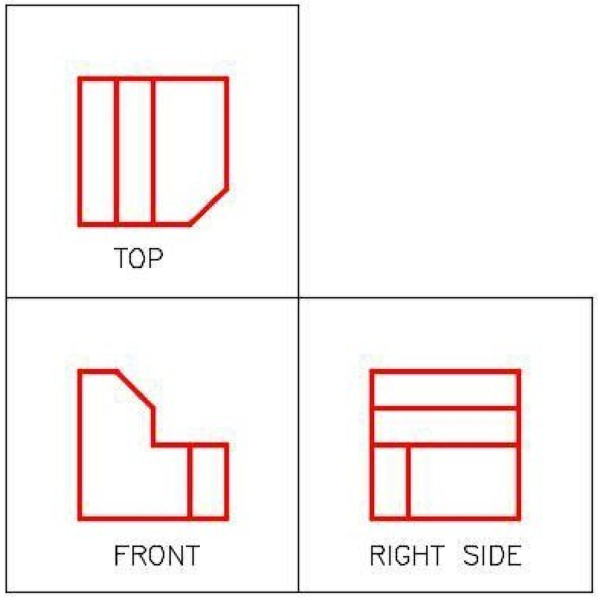


Figure 8-5
The Three Standard Views

Picking the Standard Three Views

The three standard views are always selected as shown in Figure 8-6.

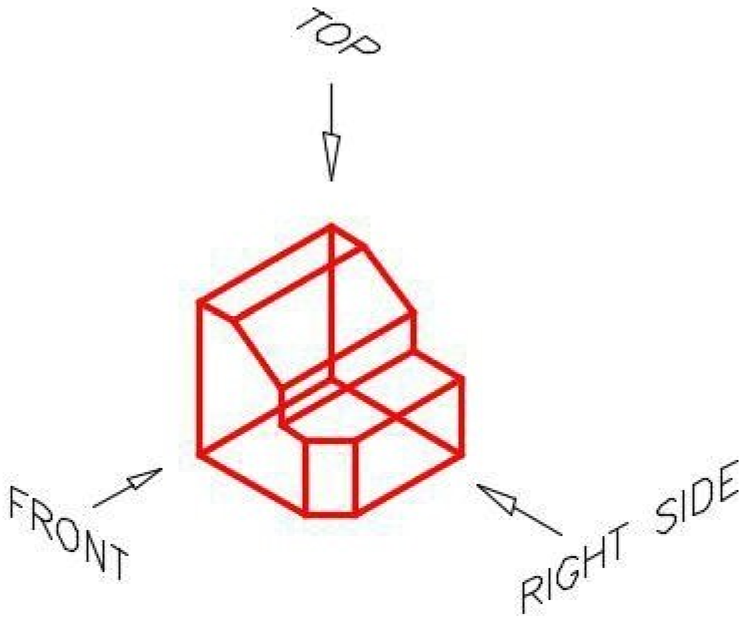


Figure 8-6
Picking the Three Standard Views

Drawing the Views

Usually it is best to draw the view with the most irregular shape first and then project lines to the other two views. For the object in Figure 8-7, the front view should be drawn first and then the top and right side views are projected. Notice how the views have to align. Figure 8-7 shows two different methods of projecting lines from the top view to the right side view or vise versa. The distance between the views is not important.

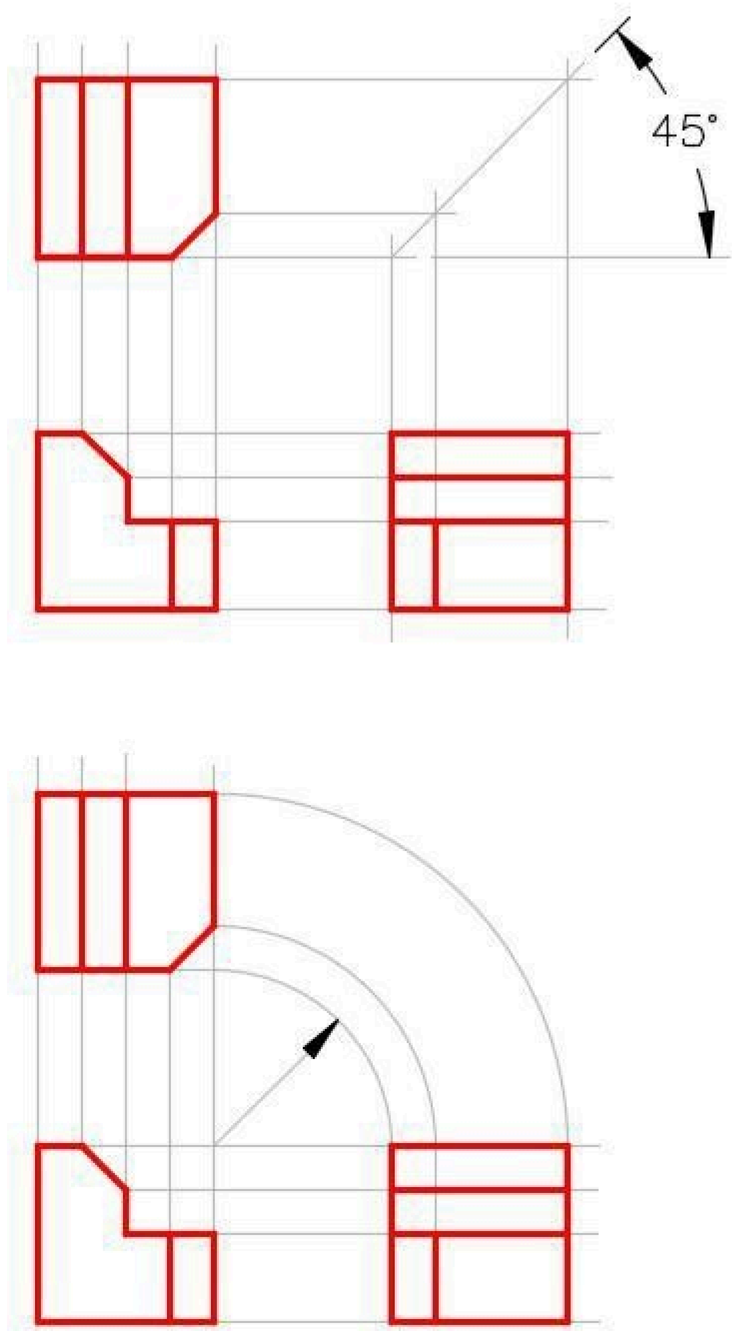


Figure 8-7
Two Methods for View Layout and Alignment

Drafting Lesson Object and Hidden Lines

Lines and features that can be seen in the views are drawn with continuous or solid lines. They are called *object lines*. Even though they are called object lines, they can be circular in shape. To completely describe an object in a multiview drawing, the drafter must also show all lines or features that are hidden in that view. They are called *hidden lines* and their linetype is dashed. Study the multiview drawing below and take note how the holes going through the object are shown with hidden lines. See Figures 8-8 and 8-9.

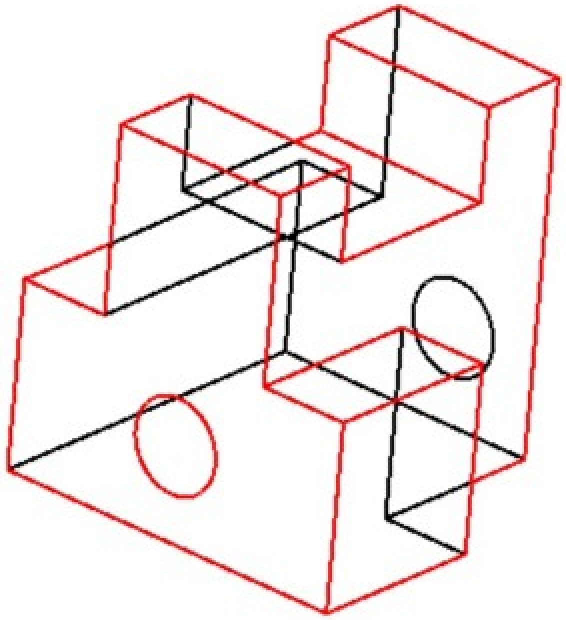


Figure 8-8
The Model

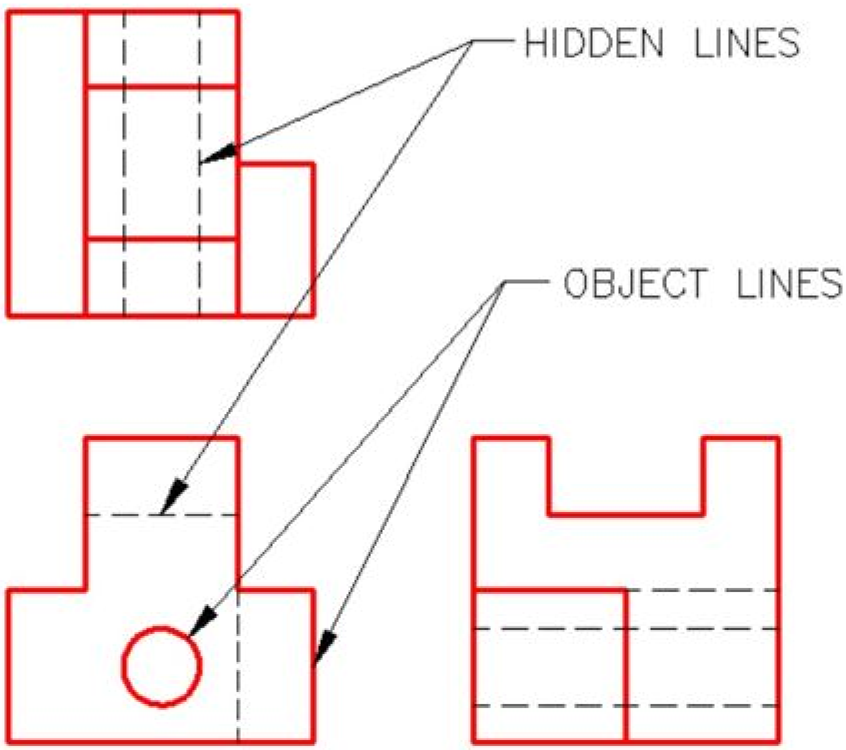


Figure 8-9
The Multiview Drawing of the Model

Key Principles

Key Principles in Module 8

1. The drafting and design world uses a system of representing a three-dimensional object by drawing two-dimensional This is called a multiview drawing.
2. The three standard views of a multiview drawing are the Top, Front, and Right Side.
3. When drawing a multiview drawing, the three standard views of an object must be drawn in the correct position and must be aligned. The distance between the views is not important.

Lab Exercise 8-1

Time allowed: 15 minutes.

Step 1

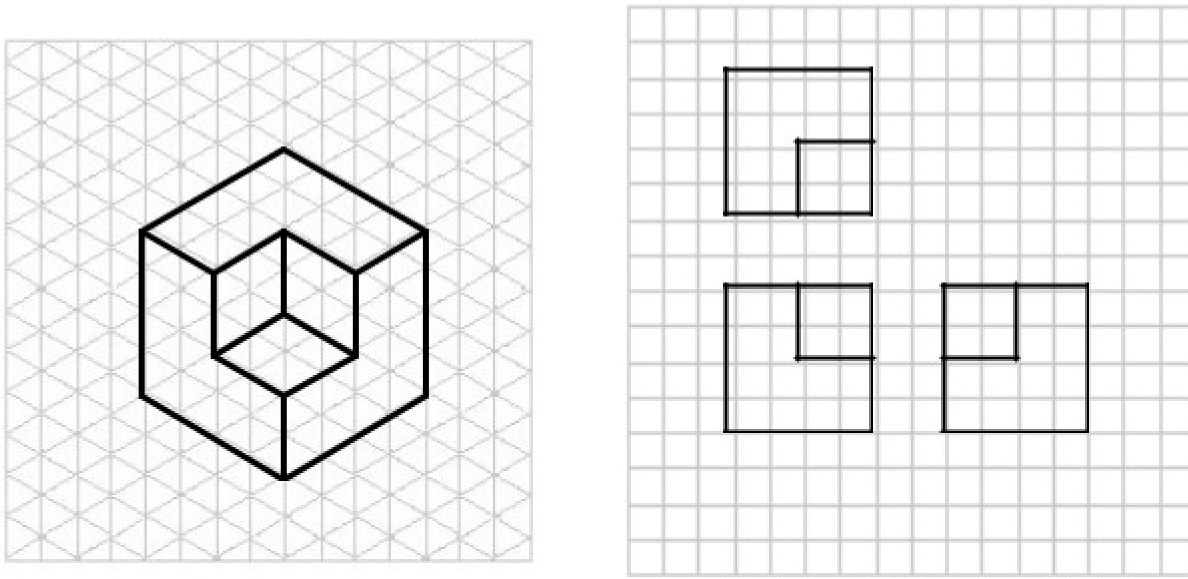
Sketch the Top, Front and Right Side views of the Object 8-1.

Step 2

Use one grid on the model equal to one grid on the drawing. See the example.

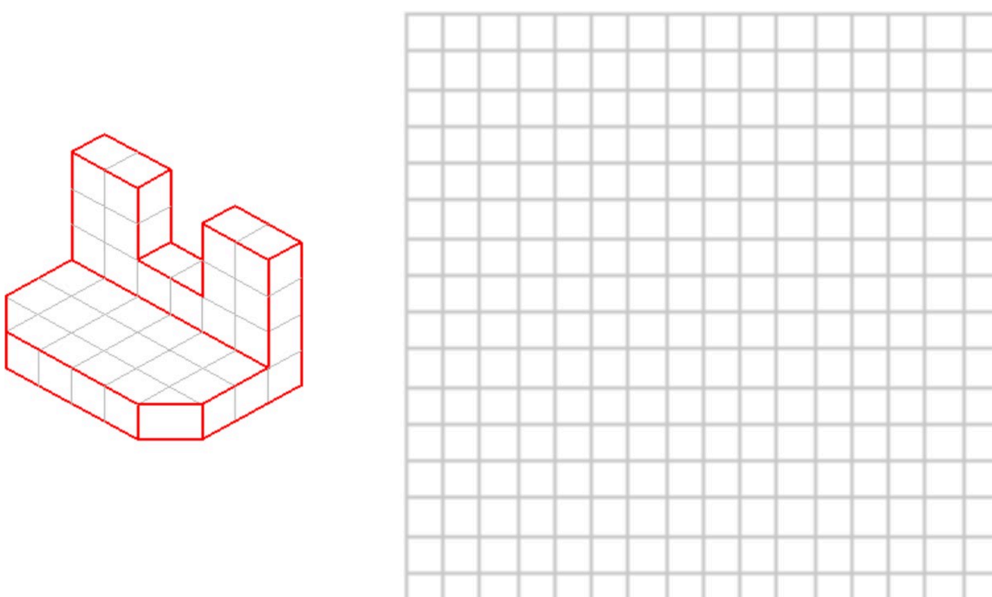
Step 3

When complete, check your answers at the end of this chapter.



Example

AUTHOR'S COMMENTS: Do this lab using paper and pencil. To print a paper copy of the graph paper on your printer, you can configure and print it free of charge using the website [www.printfreegraphpaper.com.](http://www.printfreegraphpaper.com) Set the graph paper to 1/4″ Cartesian graph paper.



Object 8-1

Lab Exercise 8-2

Time allowed: 60 minutes.

Step 1

Sketch the Top, Front, and Right Side views of each model.

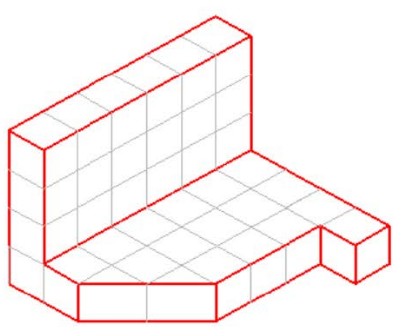
Step 2

Use one grid on the model equal to one grid on the drawing.

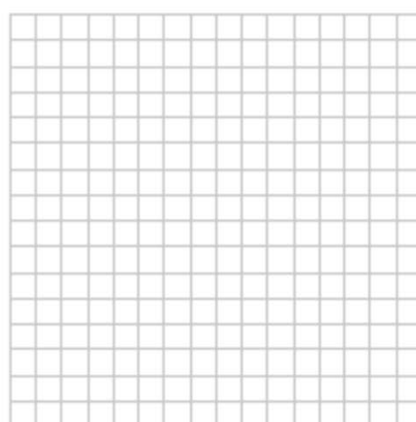
Step 3

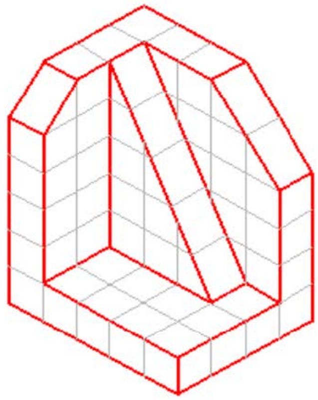
Check your answers at the end of this chapter. Do not look at the answers until you have completed your sketch.

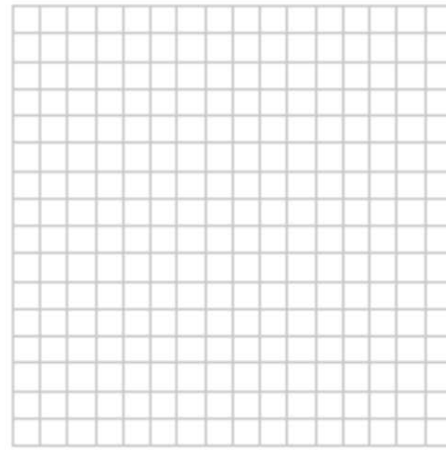
AUTHOR'S COMMENTS: Do this lab using paper and pencil. To print a paper copy of the graph paper on your printer, you can configure and print it free of charge using the website www.printfreegraphpaper.com. Set the graph paper to 1/4″ Cartesian graph paper.



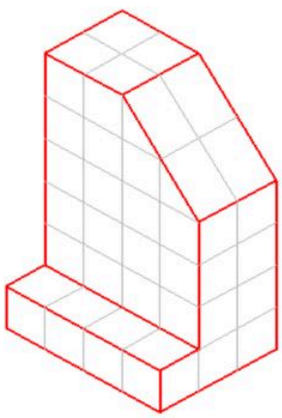
Object 8-2

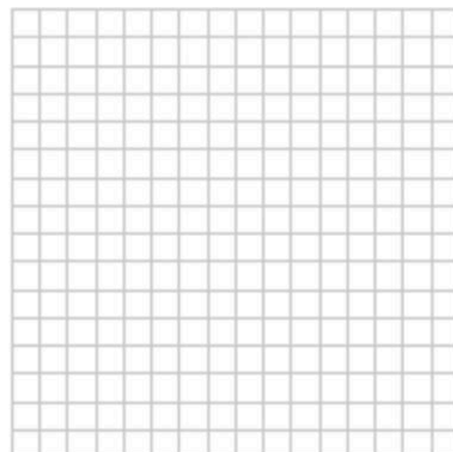




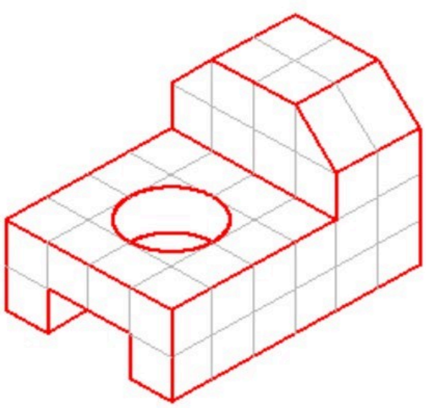


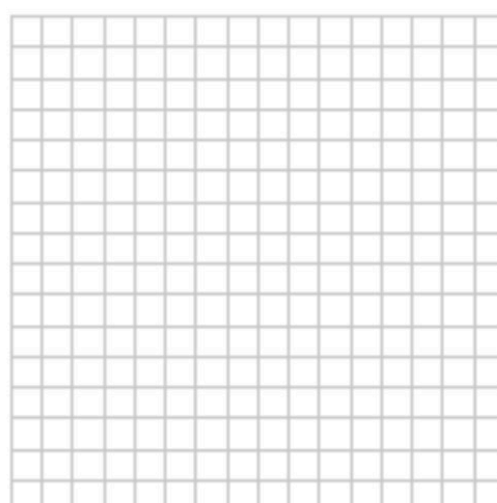
Object 8-3



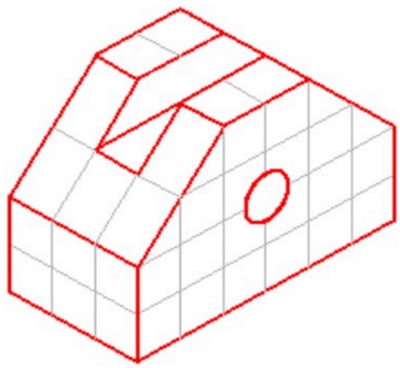


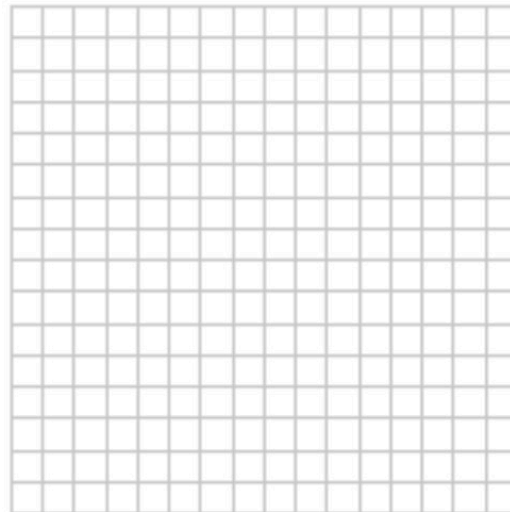
Object 8-4



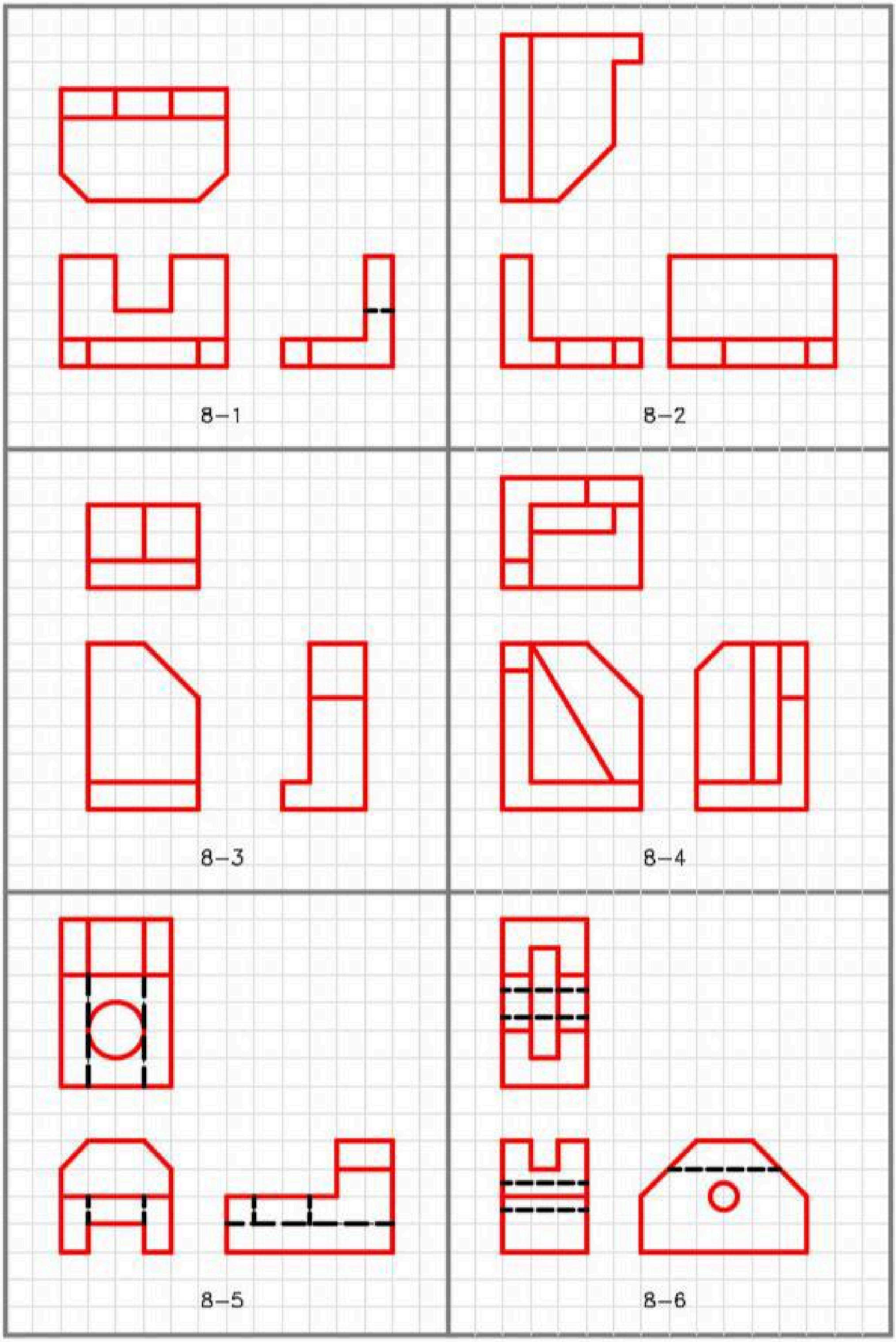


Object 8-5





Object 8-6


8–1
8–2
8–3
8–4
8–5
8–6

Module 9 Visualizing 3D Models

Learning Outcomes

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

1. Using multiview drawings, visualize and sketch isometric drawings of 3D models on isometric grid paper.

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.

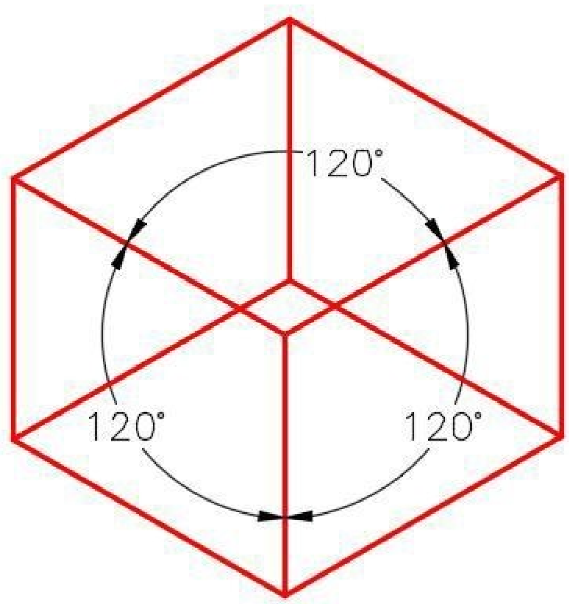


Figure 9-1
An Isometric Drawing

An *isometric drawing* is a 2-dimensional drawing that has the XYZ axis drawn at 120 degrees apart as shown in Figure 9-1. 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 9-3 shows a rectangular box drawn on the isometric grid.

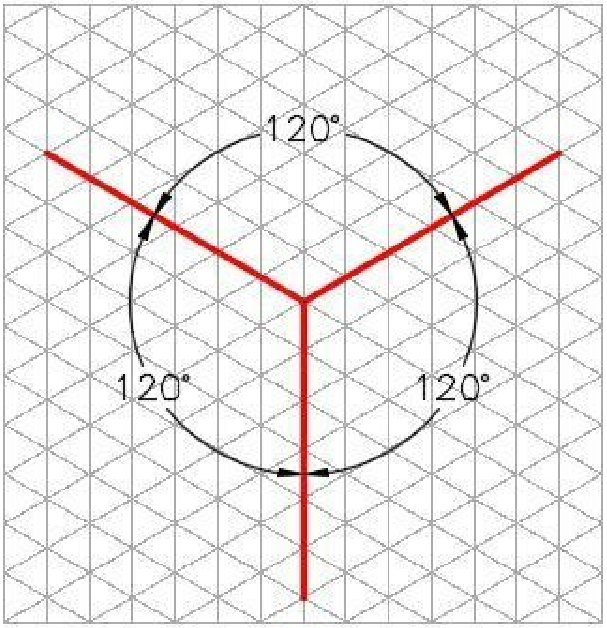


Figure 9-2
An Isometric Grid

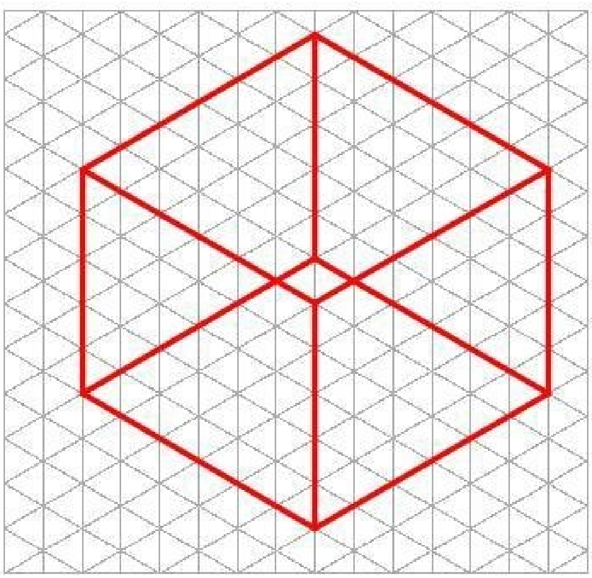


Figure 9-3
A Rectangular Box Drawn on an Isometric Grid

WORK ALONG: Sketching an Isometric Drawing of a 3D Model – Part 1

This workalong demonstrates how the multiview drawing of the 3D model, shown in Figure 9-4, is drawn as an isometric drawing.

AUTHOR'S COMMENTS: One grid on the multiview drawing is equal to one grid on the isometric grid.

AUTHOR'S COMMENTS: Complete this workalong using paper and pencil. To print a paper copy of the graph paper on your printer, you can configure and print it free of charge using the website: www.printfreegraphpaper.com. Set the graph paper to 1/4″ Isometric graph paper.

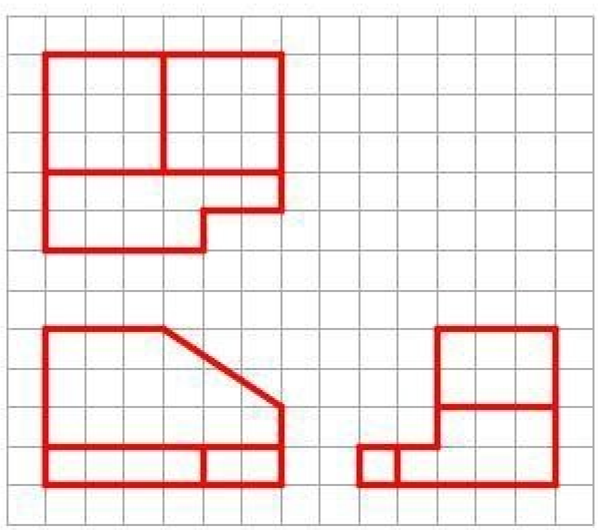


Figure 9-4
Multiview Drawing

Step 1

Using a pencil and eraser, complete this workalong to create the isometric sketch of the 3D model. (Figure Step 1)

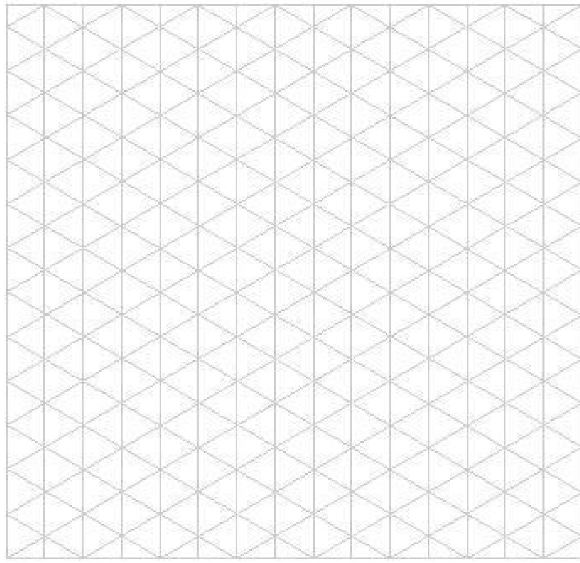


Figure Step 1

Step 2

Sketch the rectangular box using the length, width, and height of the overall size of the 3D model. The overall size of the 3D model is 6 grids long, 5 grids wide and 5 grids high. (Figure Step 2)

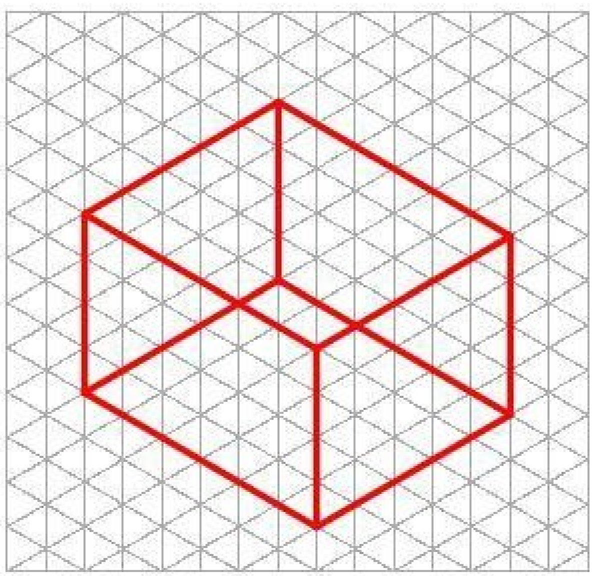


Figure Step 2

Step 3

Cut the shape away, one view at a time. Draw the Front view first. (Figure Step 3A and 3B)

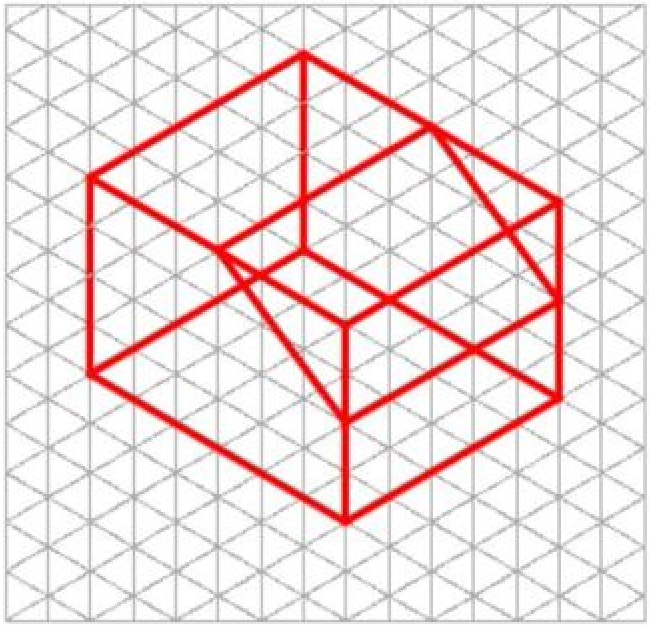


Figure Step 3A

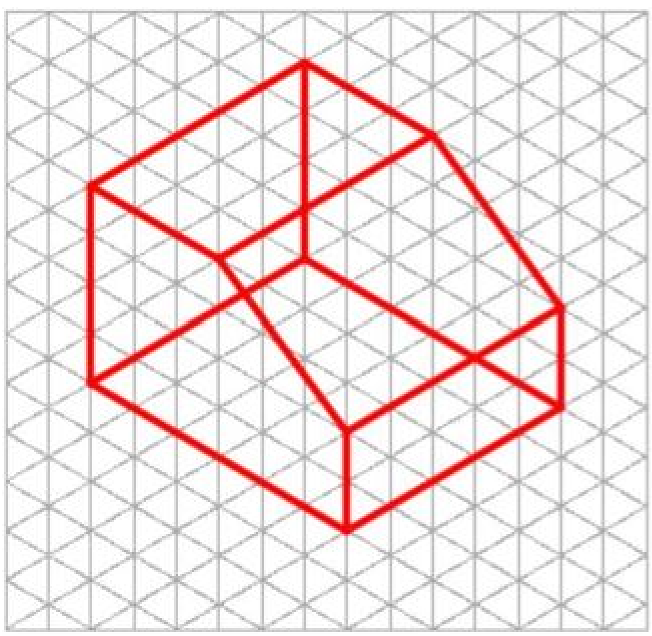


Figure Step 3B

Step 4

In the Right Side view, remove the top left side of the object to match the multiview's right side view. (Figure Step 4)

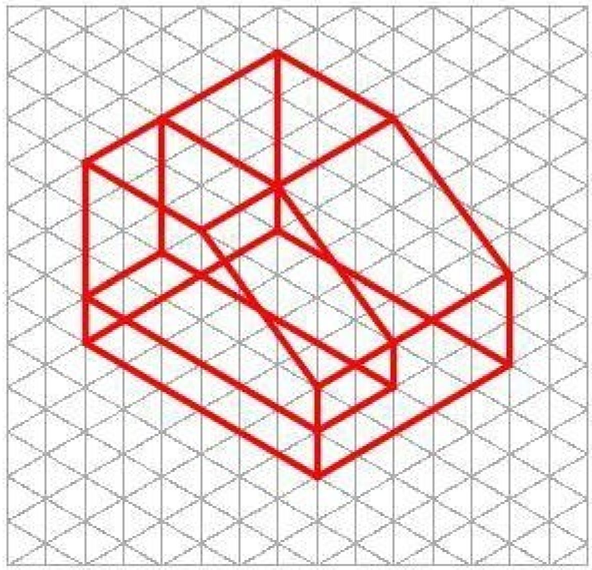


Figure Step 4

Step 5

In the Top view, remove the bottom right corner to complete the isometric drawing. (Figure Step 5A and 5B)

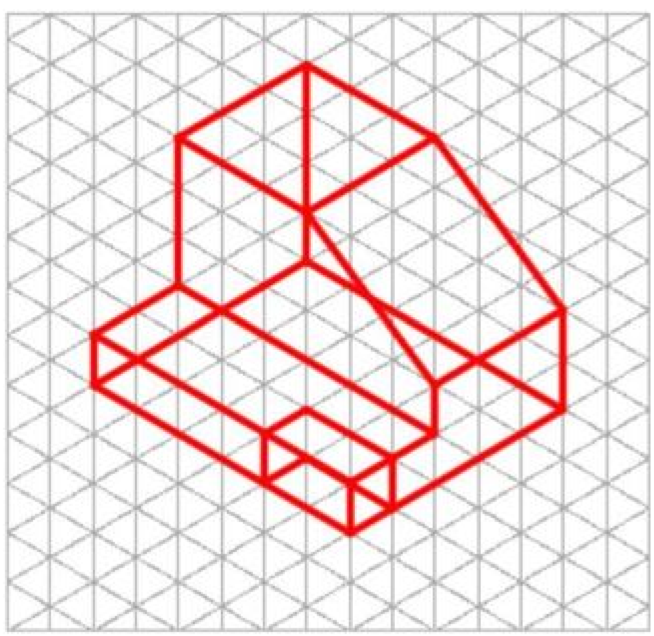


Figure Step 5A

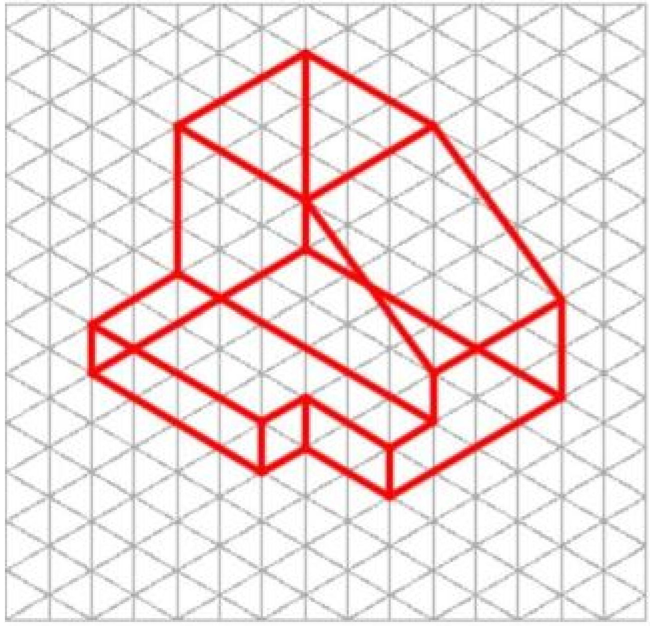


Figure Step 5B

WORK ALONG: Sketching an Isometric Drawing of a 3D Model – Part 2

This workalong demonstrates how the multiview drawing of the 3D model, shown in Figure 9-5, is drawn as an isometric drawing.

AUTHOR'S COMMENTS: One grid on the multiview drawing is equal to one grid on the isometric grid.

AUTHOR'S COMMENTS: Complete this workalong using paper and pencil. To print a paper copy of the graph paper on your printer, you can configure and print it free of charge using the website: www.printfreegraphpaper.com. Set the graph paper to 1/4″ Isometric graph paper.

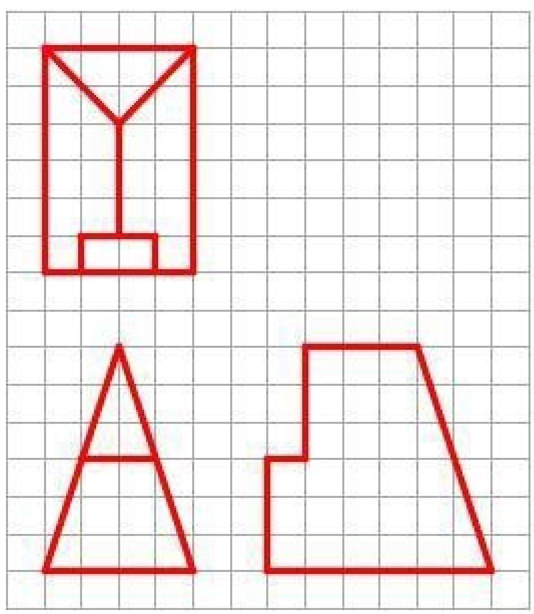


Figure 9-5
Multiview Drawing

Step 1

Using a pencil and eraser, complete this workalong to create the isometric sketch of the 3D model. (Figure Step 1)

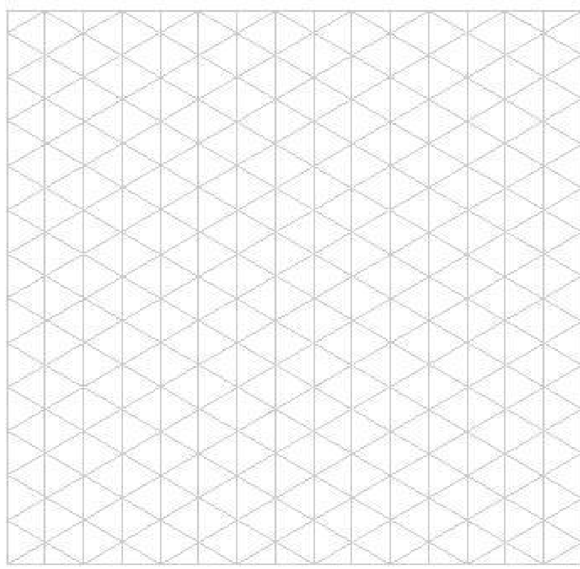


Figure Step 1

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 2

The figures show the necessary steps. Try to complete the isometric without looking at the figure. (Figure Step 2A, 2B, 2C, and 2D)

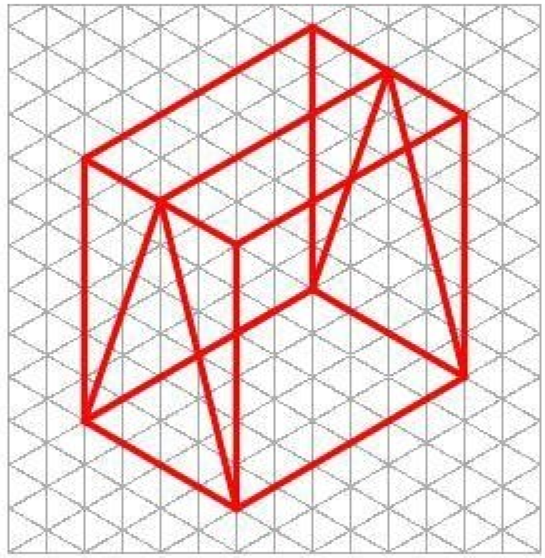


Figure Step 2A

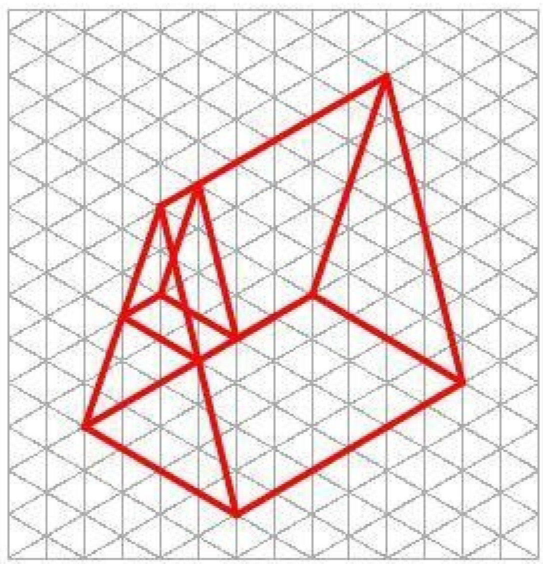


Figure Step 2B

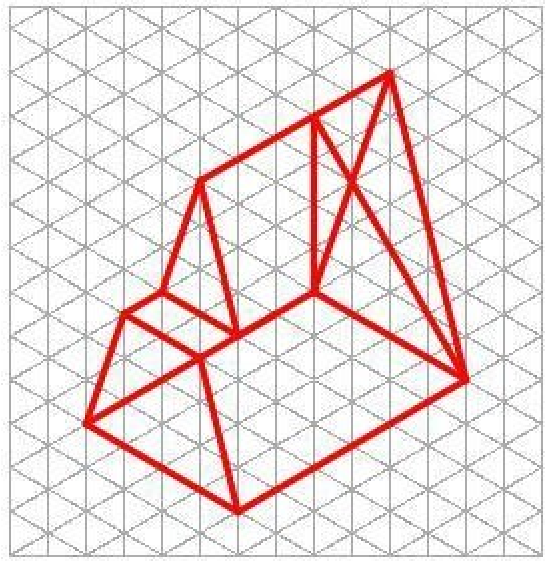


Figure Step 2C

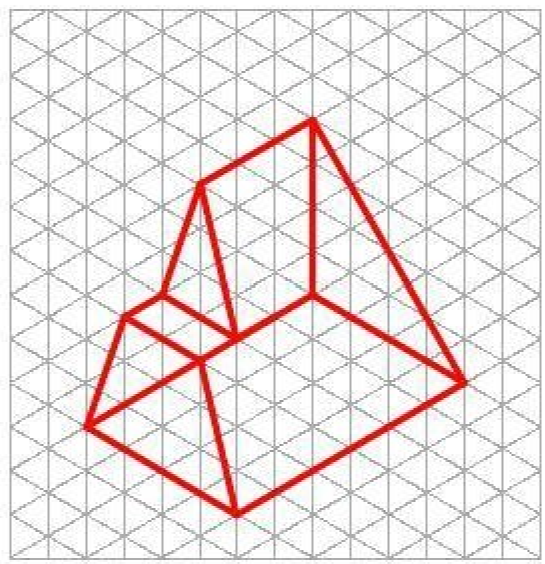


Figure Step 2D

Key Principles

Key Principles in Module 9

1. 1 An isometric drawing is a 2-dimensional drawing that has the XYZ axis drawn 120 degrees apart.
2. To sketch an isometric circle, first consider which one of the three isometric planes it is located on.

Lab Exercise 9-1

Time allowed: 90 minutes.

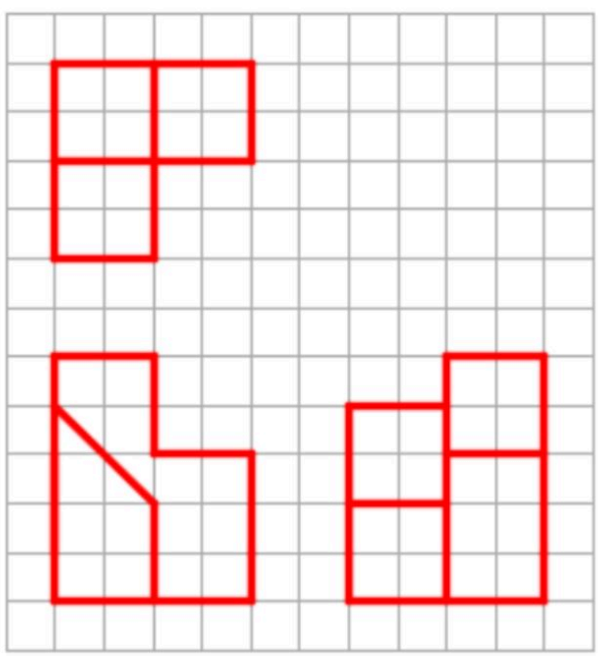
AUTHOR'S COMMENTS: Complete this workalong using paper and pencil. To print a paper copy of the graph paper on your printer, you can configure and print it free of charge using the website: www.printfreegraphpaper.com. Set the graph paper to 1/4″ Isometric graph paper.

Step 1

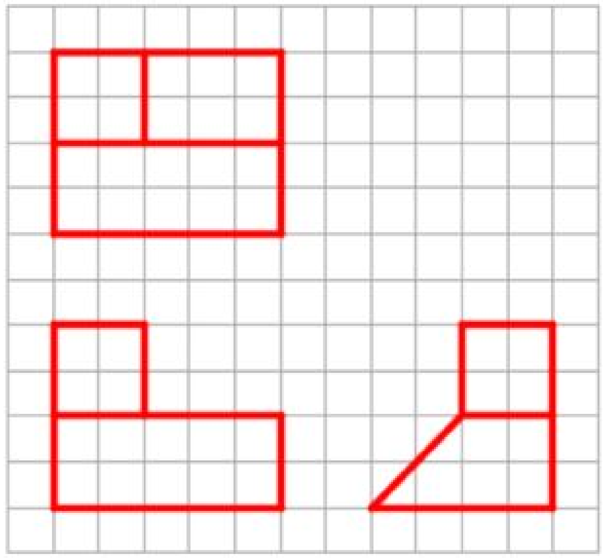
Using the 4 multiview drawings shown below, using pencil and eraser and the graph paper you printed, sketch the isometric drawing of each 3D model.

Step 2

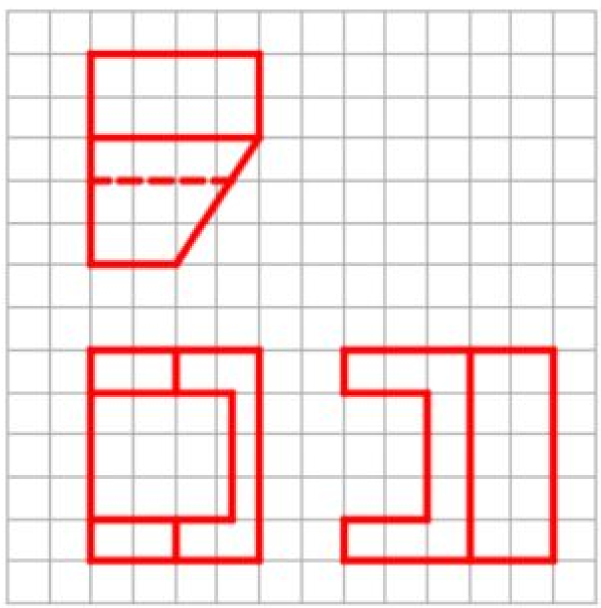
Using what was just taught, draw isometric drawings of the four objects: Object 9-1 to Object 9-4. For the answers, see the end of this chapter. Try to visualize the 3D model by looking at the multiview drawing and then draw the isometric. Do not look at the answers until you have done your best to complete the isometric drawing of each object.



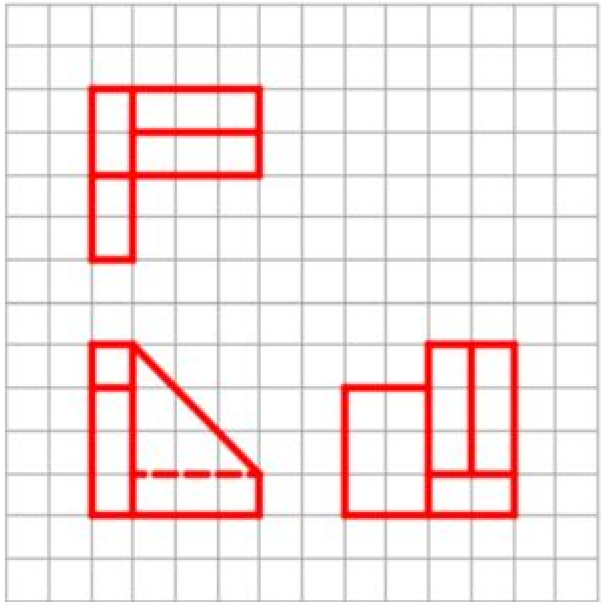
Object 9-1



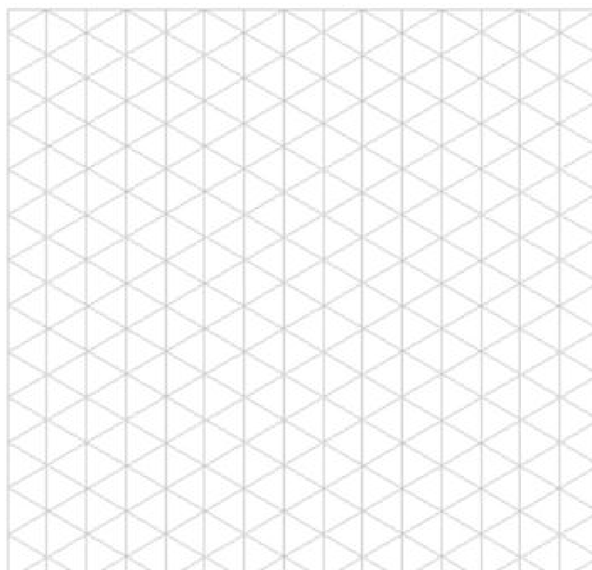
Object 9-2



Object 9-3

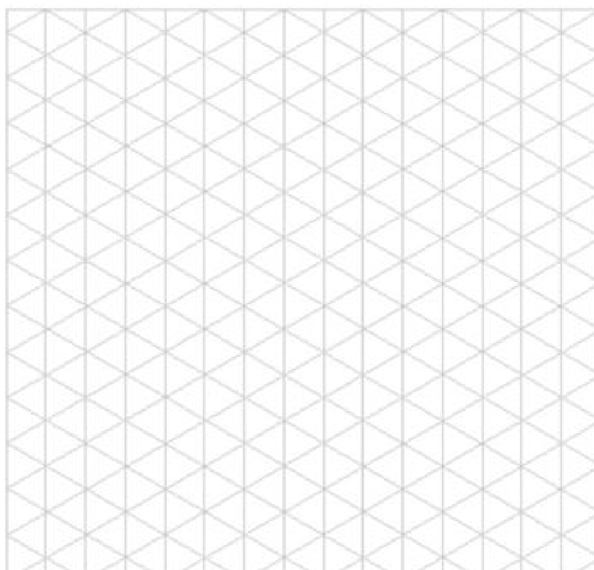


Object 9-4

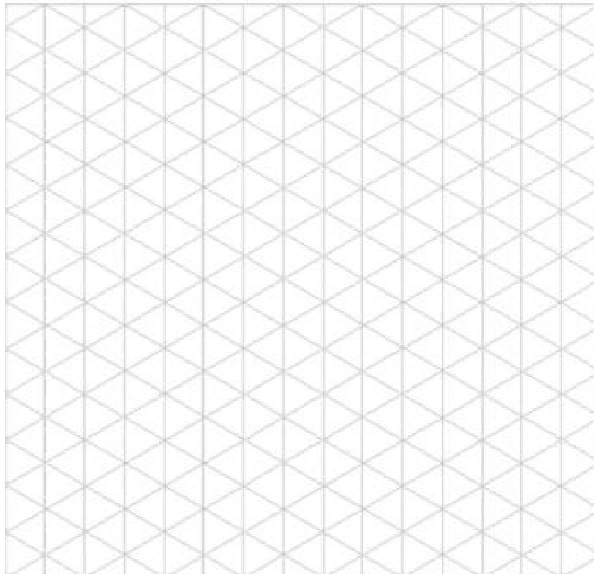


Object 9-1

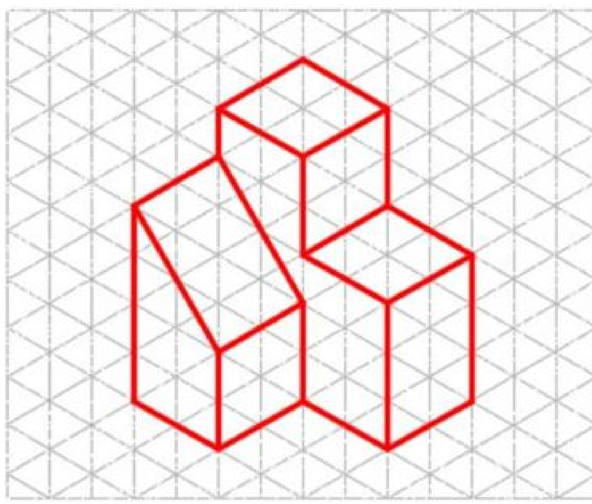
Object 9-2



Object 9-3

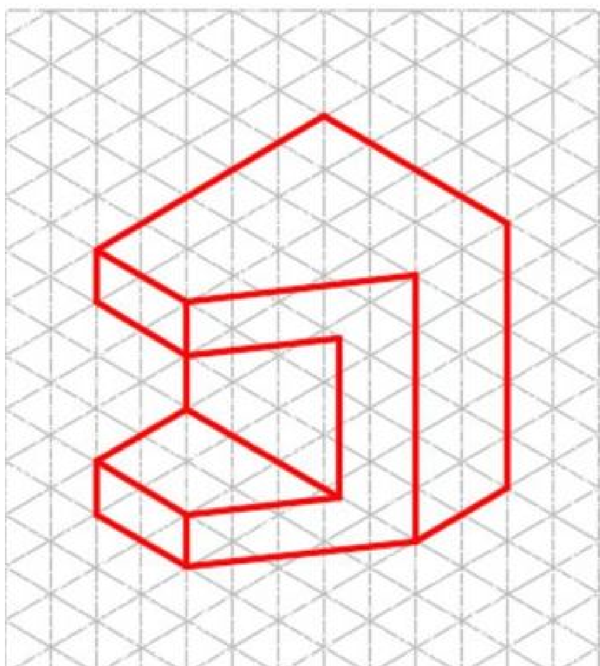


Object 9-4

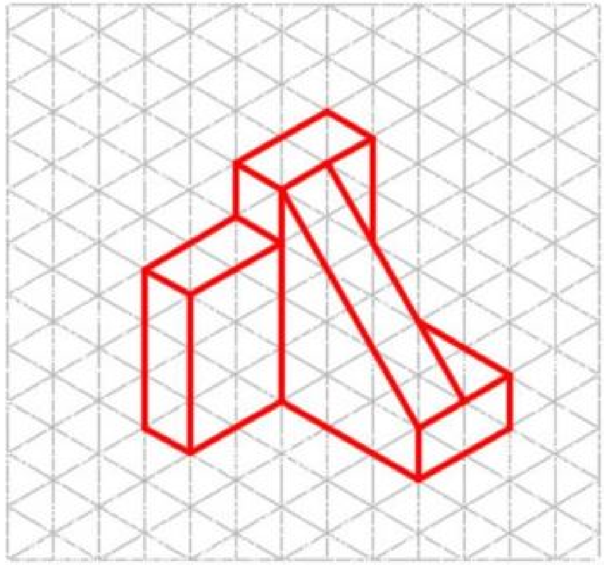


Object 9-1

Object 9-2



Object 9-3



Object 9-4

Module 10 2D Sketching Planes

Learning Outcomes

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

1. Describe the three predefined 2D sketching planes and the view of the model that they are on.
2. Construct solid models by drawing the Base sketch on either the Front or Right Side instead of the default Top view.
3. Describe a consumed and an unconsumed sketch.

2D Sketching Planes

Up to this point in the book, the Base sketch has been drawn on the XY plane. The XY plane is the Top view of the model and the default plane as configured in the templates that are being used to complete the workalongs and lab exercises in the Inventor book. The models that have been constructed up to this point in the modules were all designed so that the Base sketch was drawn on the XY plane or the Top view. In this module, learning how to construct solid models by drawing the Base sketch on either the front or right side planes will be taught.

Inventor has p three predefined planes that can be used to draw the Base sketch. They are the XY, XZ, and YZ planes. The XY plane is the Top view, the XZ plane is the Front view and the YZ is the Right Side view of the model.

Keep in mind the rule that was taught in Module 4. ‘ It is best to draw the Base sketch on the plane that has the most complex contour. Contours with arcs and curves should be avoided ‘.

The Three Predefined Planes

To help visualize the *three predefined planes* used in Inventor, the 3D model shown in Figure 10-1 is used in this module. The glass box principle that was taught in Module 8 is used to help you visualize Inventor's three predefined planes. See Figure 10-2.

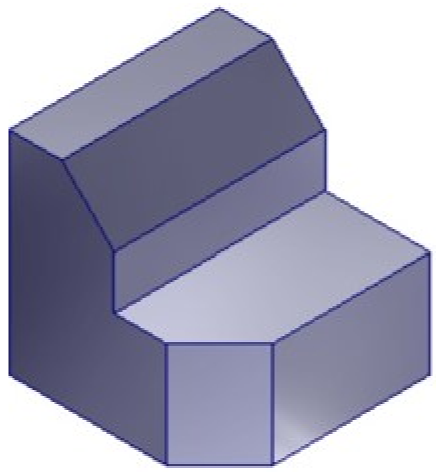


Figure 10-1
The 3D Model

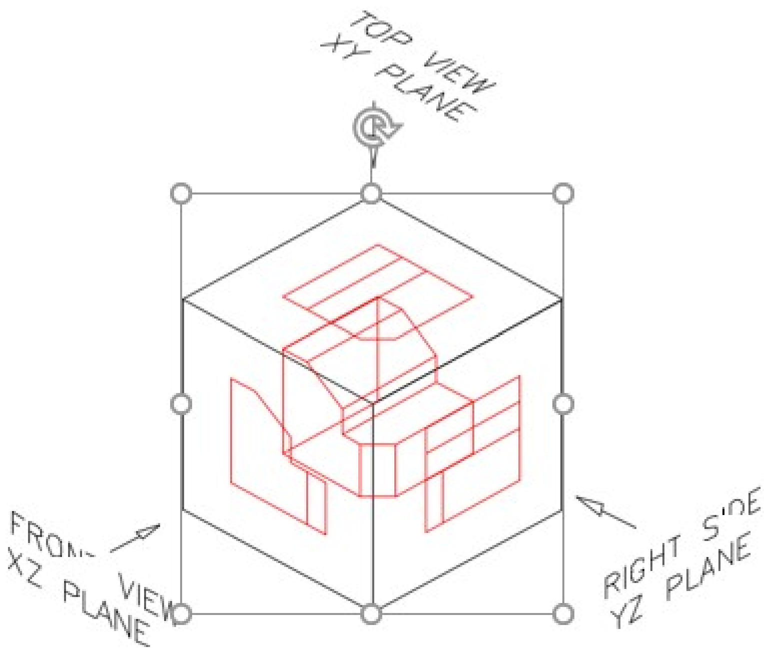


Figure 10-2 The Glass Box

Consumed and Unconsumed Sketches

A *consumed sketch* is a 2D sketch that has been extruded or revolved to create a 3D solid model. An *unconsumed sketch* is a 2D sketch that is blank or one that has not been extruded or revolved.

The Browser bar will display which sketches have been consumed and which ones are unconsumed. See Figure 10-3. Sketch1 is unconsumed while Sketch2, Sketch3, and Sketch4 have been extruded and are consumed.

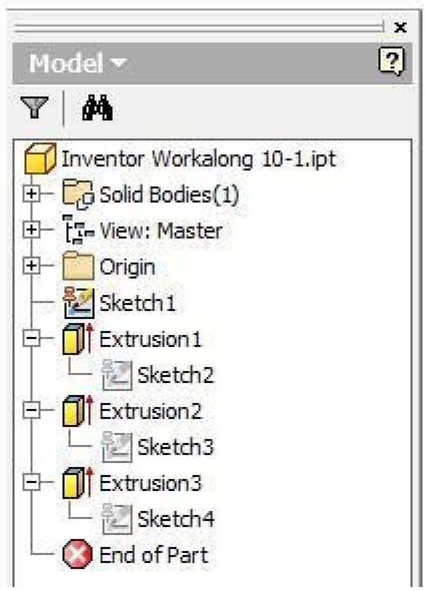
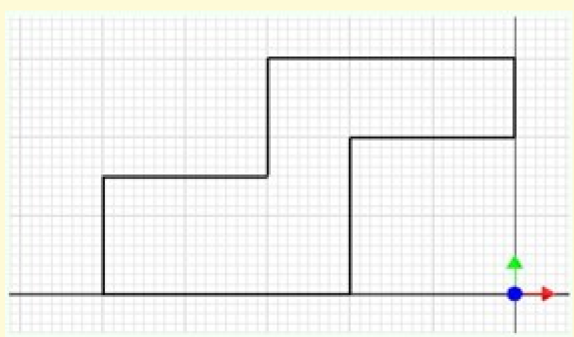
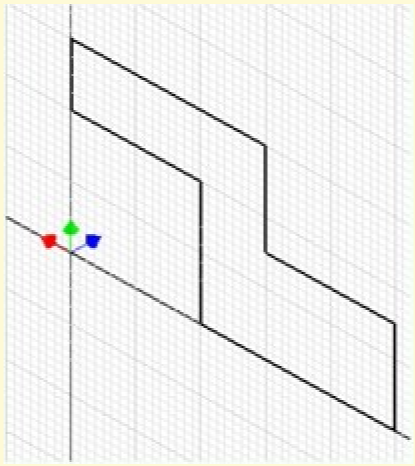


Figure 10-3
Consumed and Unconsumed Sketches

USER TIP: A 2D sketch can be drawn in the orthographic view or in the 3D Home view. In fact, any 3D orbited view can be used. When drawing in a 3D view, it is always best to draw in the Home view since this helps you maintain a good mental picture of the model.



2D Orthographic View



3D Home View

MUST KNOW: Inventor has three predefined planes that can be used to draw the Base sketch on. They are the XY, XZ and YZ planes. The XY plane is the Top view, the XZ plane is the Front view and the YZ is the Right Side view of the Base model.

WORK ALONG: Working with 2D Sketching Planes

Step 1

Start a new part file using the template: English-Modules Part (in).ipt.

Step 2

In Sketch mode, press F6 to change to the Home view. The Graphic window and the Browser bar will appear as shown in the figure. (Figure Step 2)

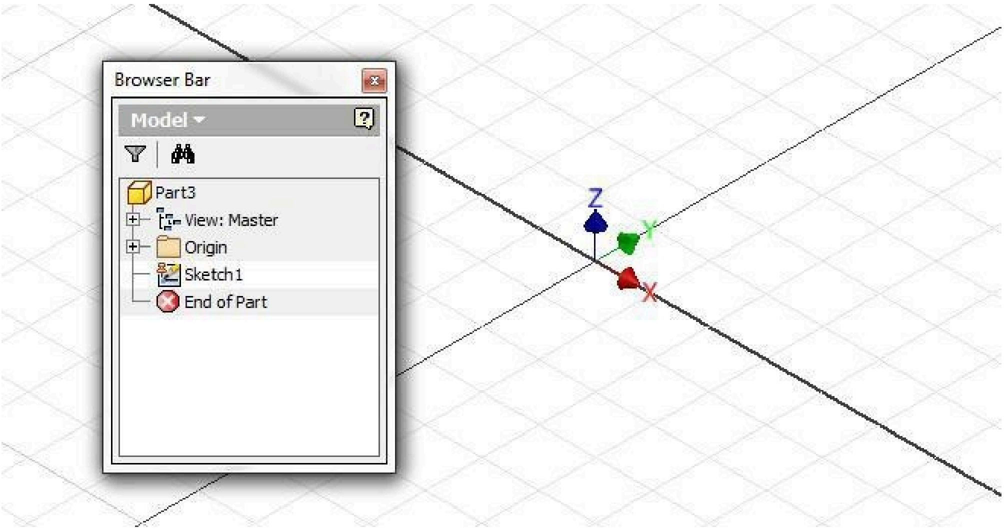


Figure Step 2 [Click to see image full size]

AUTHOR'S COMMENTS: This is the Top view or the XY plane since that is Inventor's default plane. Note that Sketch1 is always on the default plane.

Step 3

In the Browser bar, expand the folder: Origin as shown in the figure. Place the cursor on the XY Plane. Note the orientation of the plane on the sketch. (Figure Step 3)

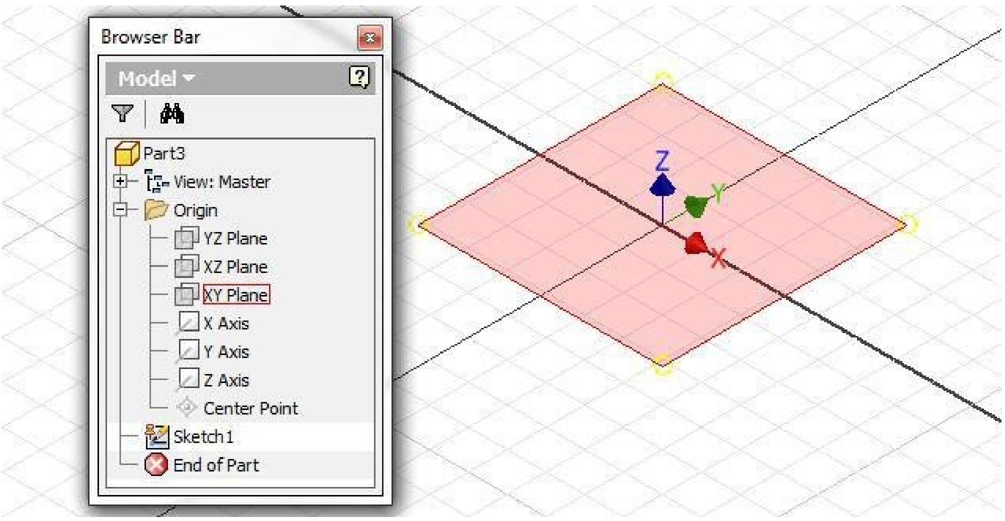


Figure Step 3 [Click to see image full size]

Step 4

Place the cursor on the XZ Plane in the Browser bar. Note the orientation of the plane on the sketch. The XZ Plane is the Front view. (Figure Step 4)

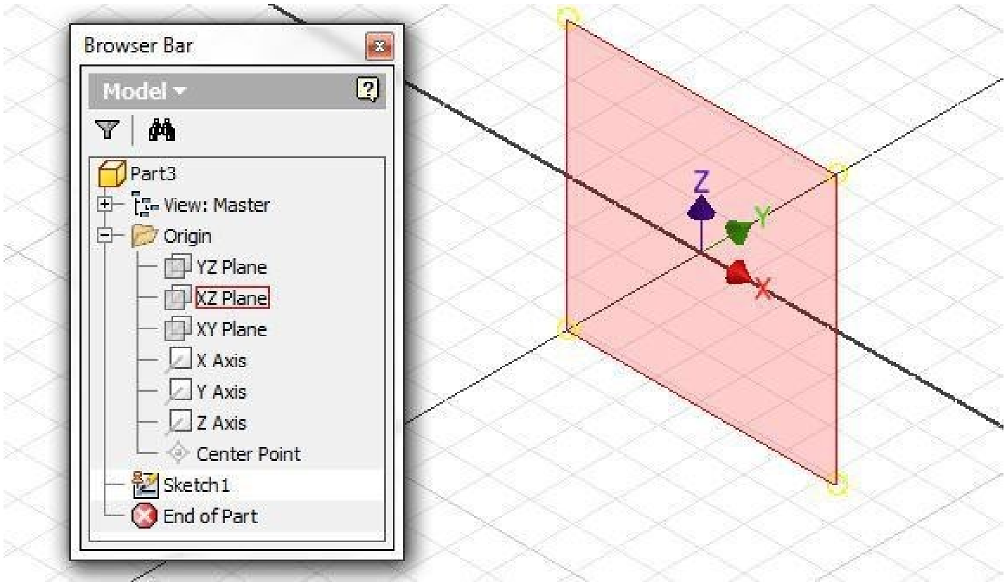


Figure Step 4 [Click to see image full size]

Step 5

Place the cursor on the YZ Plane in the Browser bar. Note the orientation of the plane on the sketch. This is the Right Side view. (Figure Step 5)

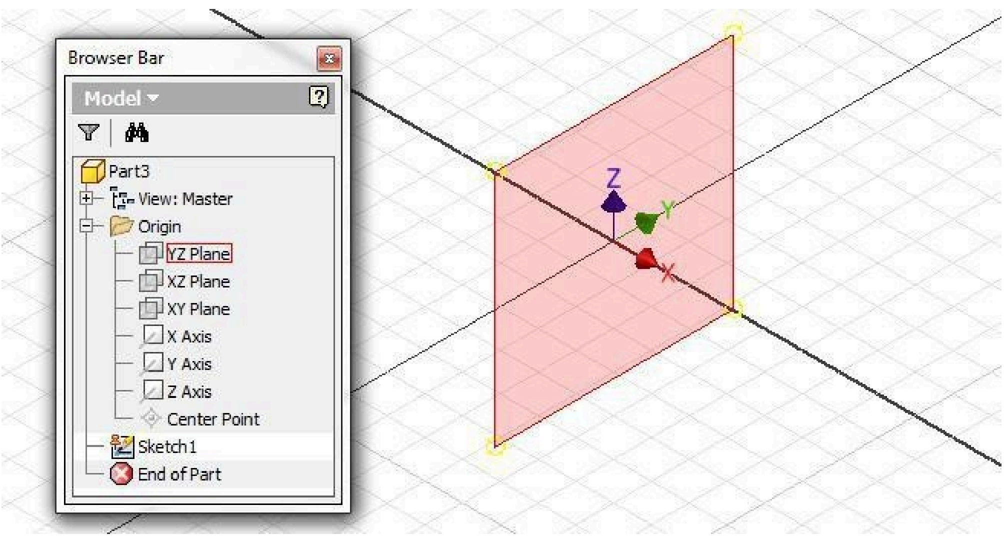


Figure Step 5 [Click to see image full size]

Step 6

Place the cursor anywhere in the Graphic window and right click the mouse. In the Right-click menu, click Finish 2D Sketch.

AUTHOR'S COMMENTS: The object shown in Figures 10-3 and 10-4 is the model that you will be constructing in this workalong.

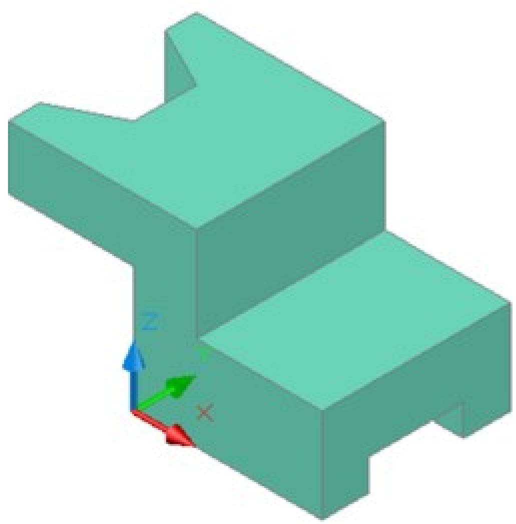


Figure 10-3
3D Model – Home View

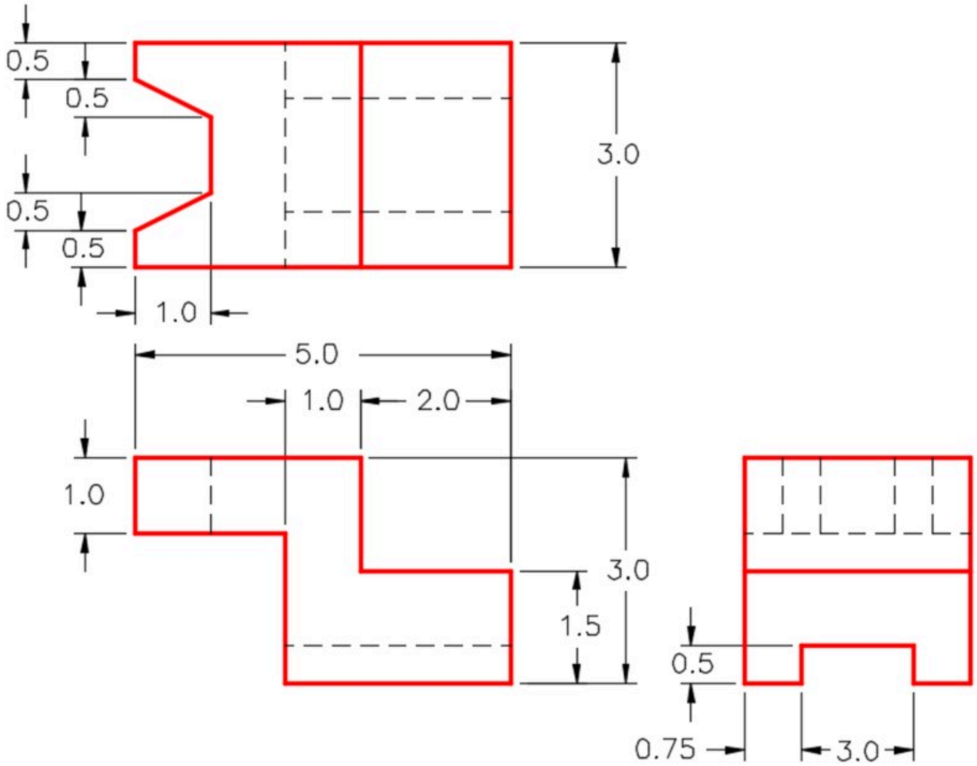


Figure 10-4
Dimensioned Multiview Drawing [Click to see image full size]

AUTHOR'S COMMENTS: Before starting the Base sketch you must pick the best view to draw it on. It should, in most cases, be the view with the most complex contour. For this model, the best view to use is the Front view or the XZ plane.

Step 7

Save the part file with the name: Inventor Workalong 10-1. In Model mode, expand the folder: Origin in the Browser bar and right-click the XZ plane. In the Right-click menu, click New Sketch as shown in figure. (Figure Step 7)

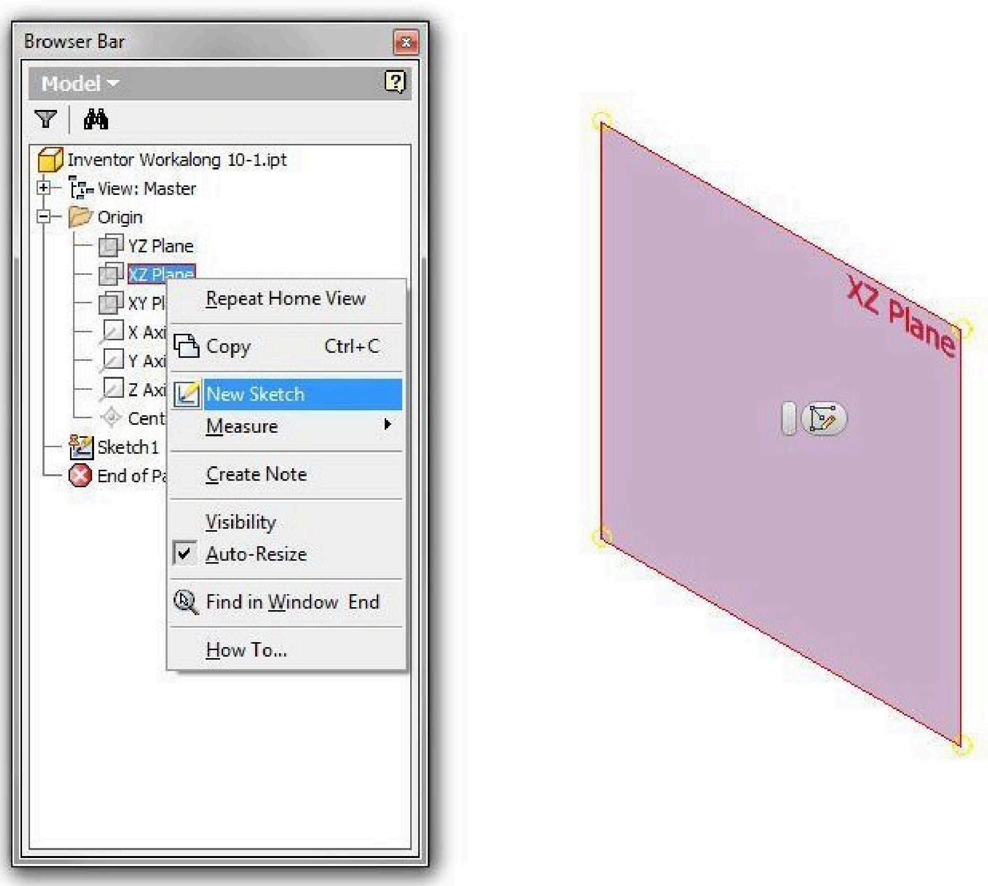


Figure Step 7 [Click to see image full size]

AUTHOR'S COMMENTS: Since Sketch1 was not used, it is blank and is an unconsumed sketch.

Step 8

The Graphic window will change to Sketch mode. Change to the Home view. Note that in the Browser bar a new sketch will appear and named Sketch2. (Figure Step 8)

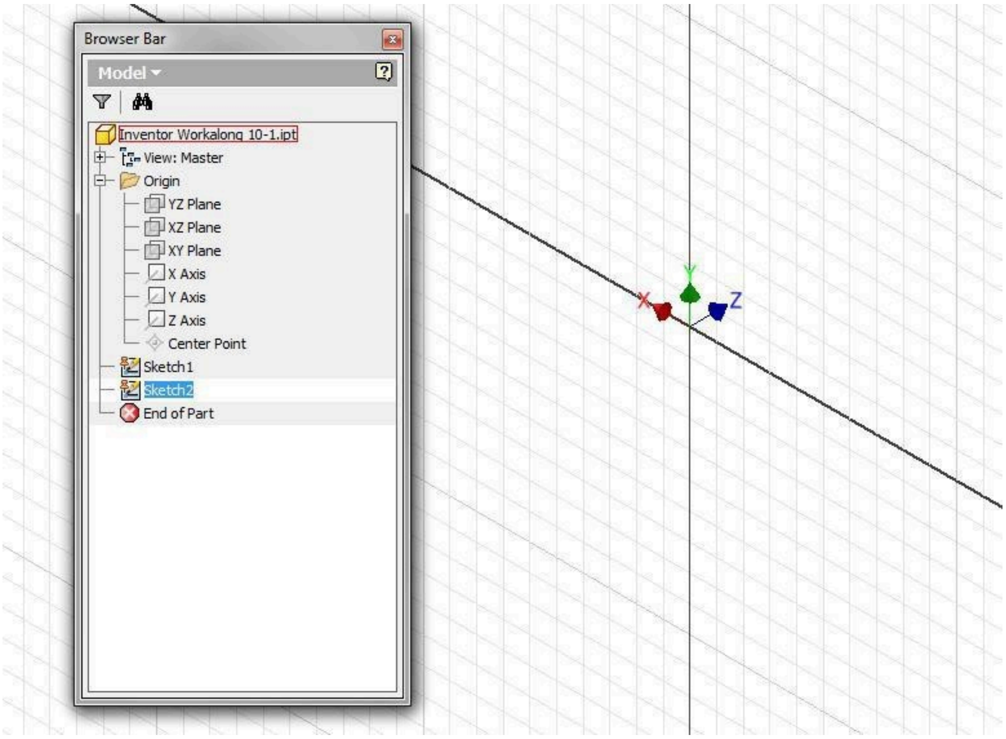


Figure Step 8 [Click to see image full size]

AUTHOR'S COMMENTS: Since Sketch1 was not used, it is blank and is an unconsumed sketch.

Step 9

Project the Center Point onto the sketch plane. Draw the Base sketch for the model applying all of the necessary geometrical constraints to maintain the shape of the sketch. Note the location of X0Y0Z0. Insert the necessary driving dimensions to fully constrain the sketch. (Figure Step 9)

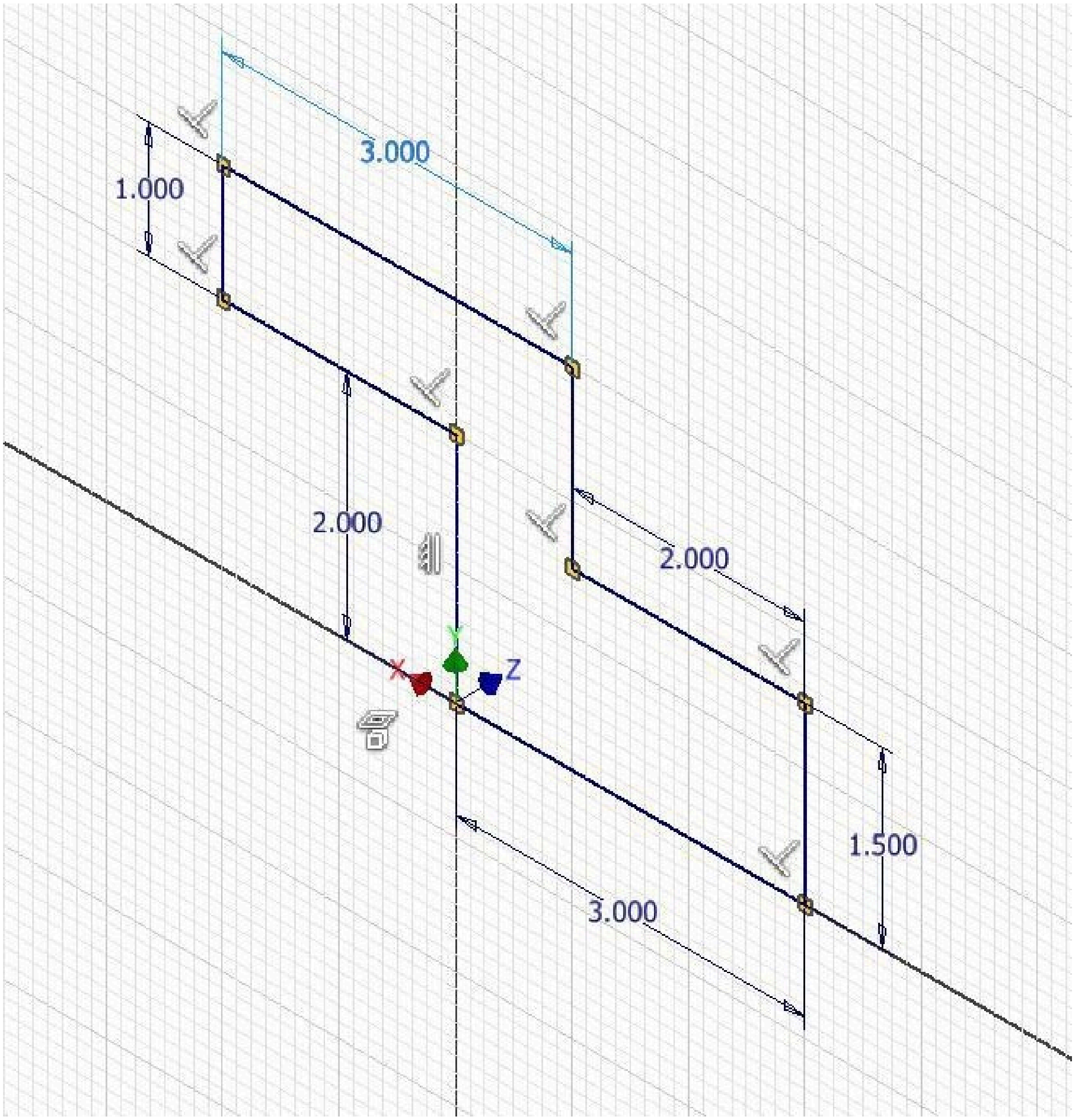


Figure Step 9 [Click to see image full size]

AUTHOR'S COMMENTS: All the lines in the sketch should display purple If they do not, start the workalong over again.

Step 10

Press PAGE UP key to execute the LOOK UP/VIEW FACE command. Select one of the lines to change the sketch to the 2D view of the the XZ plane. Press F8 to display the geometrical constraint icons. They should appear similar to the figure. (Figure Step 10)

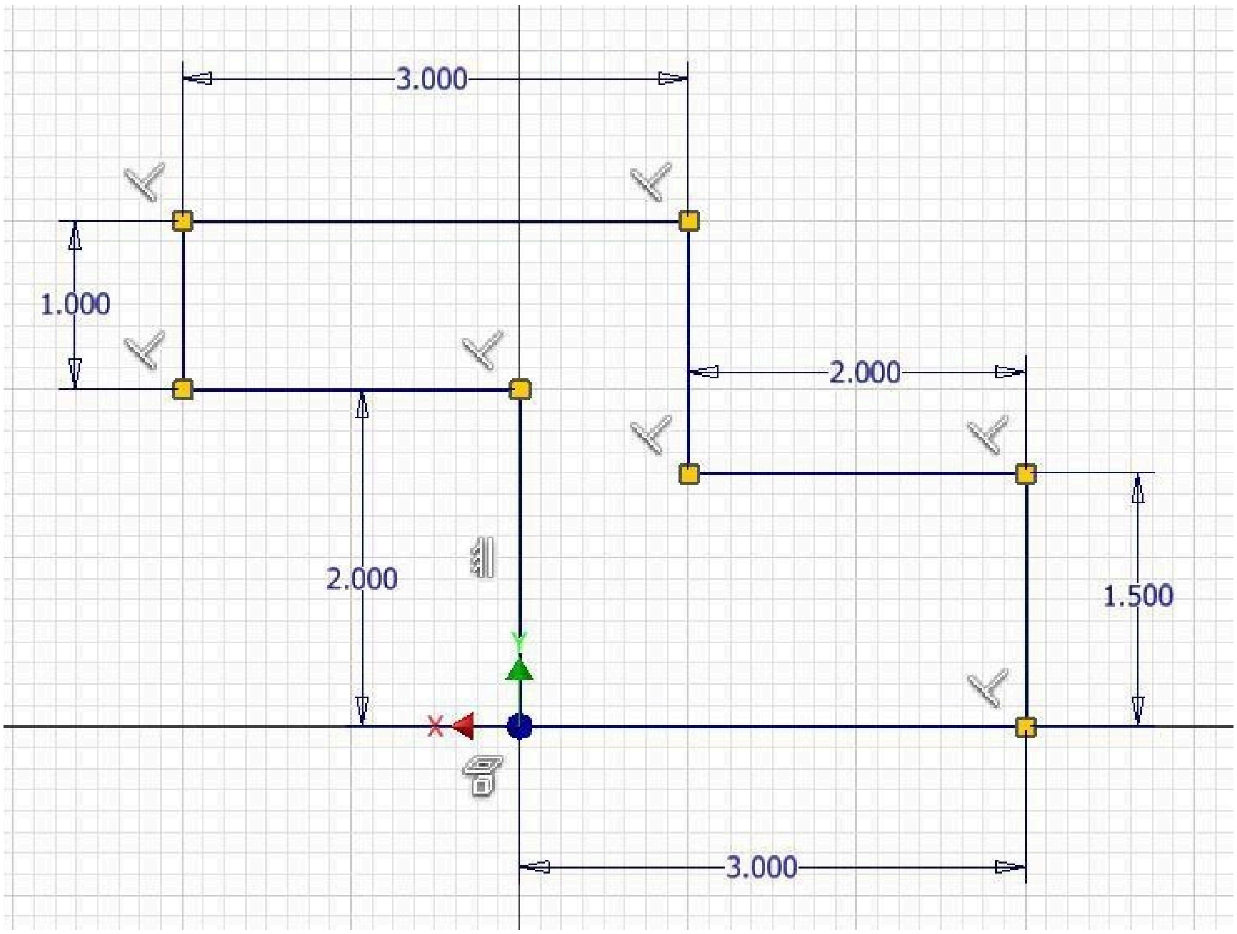


Figure Step 10 [Click to see image full size]

AUTHOR'S COMMENTS: The constraint icons in your sketch may not match the figure exactly.

Step 11

Right-click anywhere in the Graphic window. In the Right-click menu, click Finish 2D Sketch to return to Model mode. (Figure Step 11)

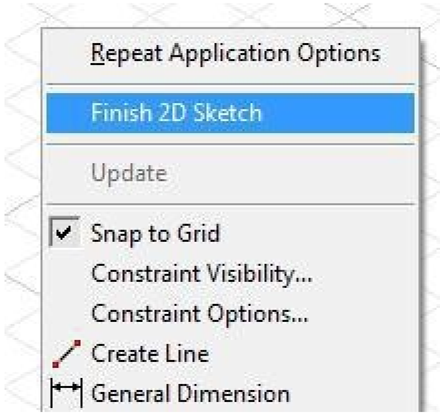


Figure Step 11

Step 12

Save the part with the name: Inventor Workalong 10-1. Extrude the sketch to create the solid model as shown in figure. (Figure Step 12)

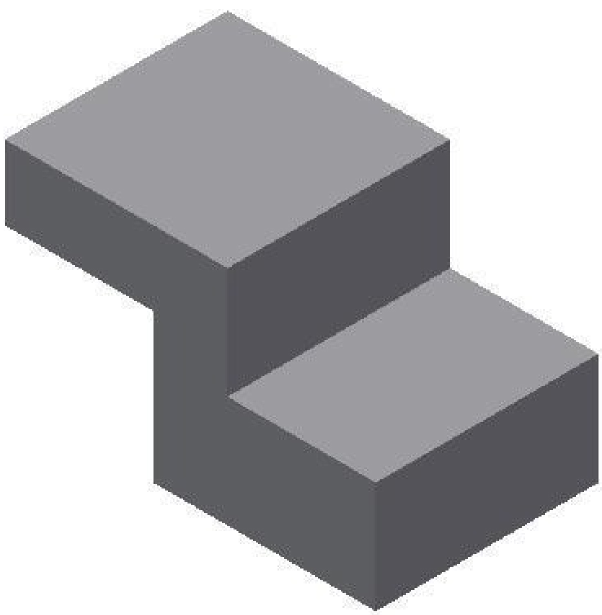


Figure Step 12

Step 13

Using the 2D SKETCH command, or even better the shortcut S, start a new sketch and select the right side as the plane to draw it on. (Figure Step 13)

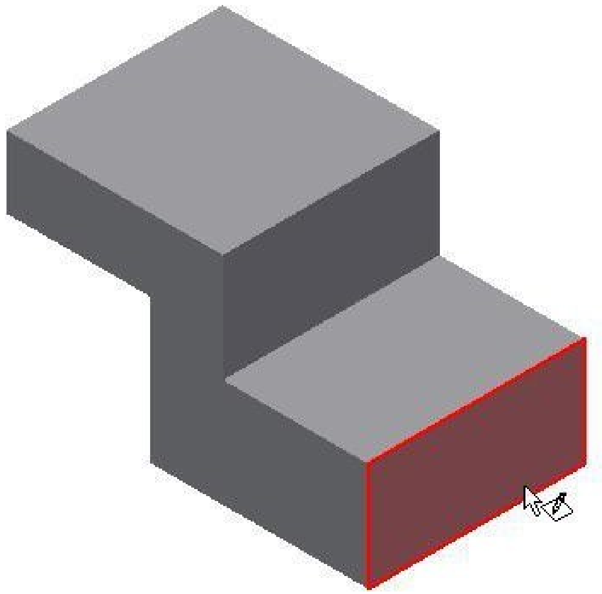


Figure Step 13

Step 14

The grid will display on the right side. It will be Sketch3 in the Browser bar. (Figure Step 14)

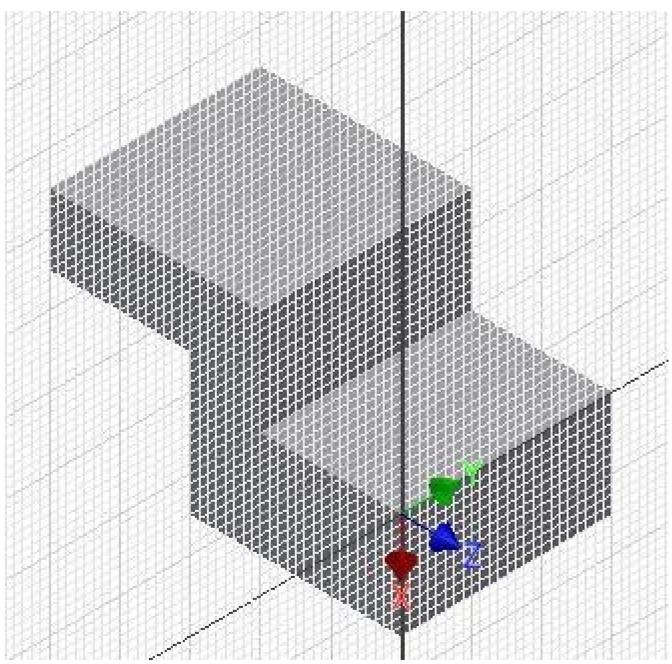


Figure Step 14

Step 15

Draw three lines for the slot. Apply all of the necessary geometrical and dimensional constraints to fully constrain the sketch. (Figure Step 15)

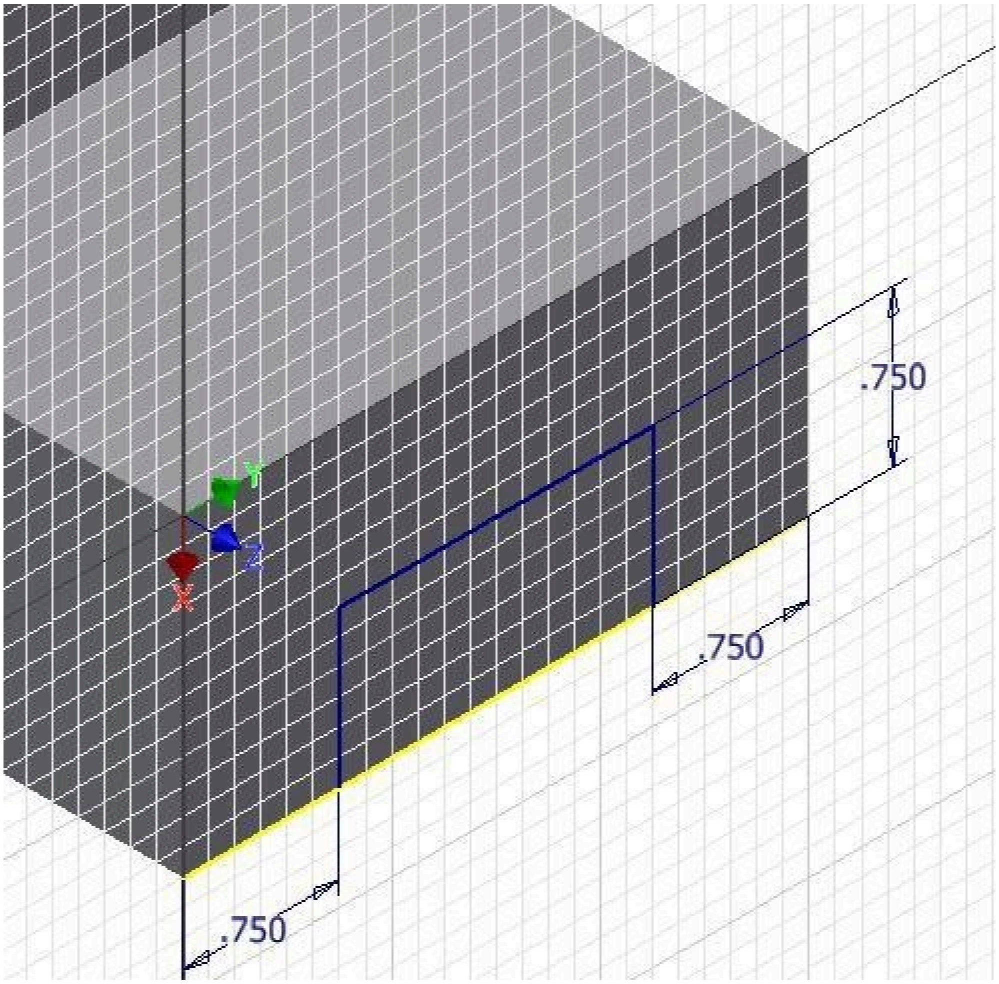


Figure Step 15 [Click to see image full size]

AUTHOR'S COMMENTS: Ensure that you snap onto the edge when you start the first line and end the third line. By doing that, you only are required to draw 3 lines to fully constrain the sketch. See Module 7 if you have trouble doing this.

AUTHOR'S COMMENTS: All three lines in the sketch should display purple to indicate the sketch is fully constrained. Do not continue on with this workalong until the sketch is fully constrained.

Step 16

Press F8 to enable the display of the constraint icons. They should be similar to the figure. (Figure Step 16)

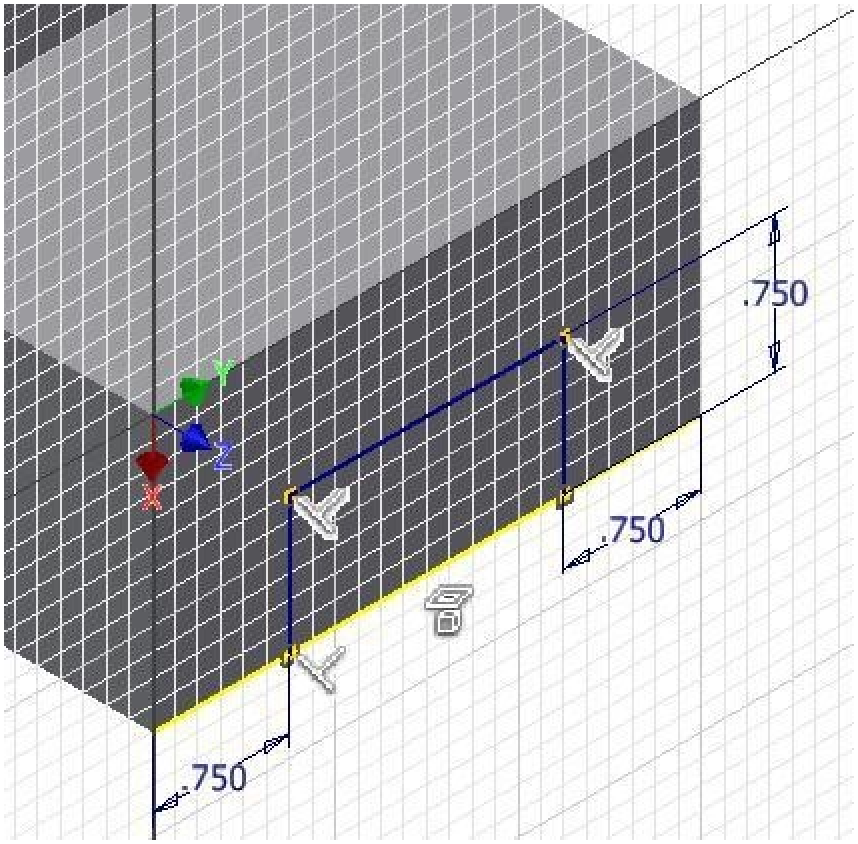


Figure Step 16

AUTHOR'S COMMENTS: Your sketch my not match the figure exactly. Ensure that your sketch fully constrained.

Step 17

Press F9 to disable the display of the constraint icons. Extrude the sketch using the To next. (Figure Step 17)

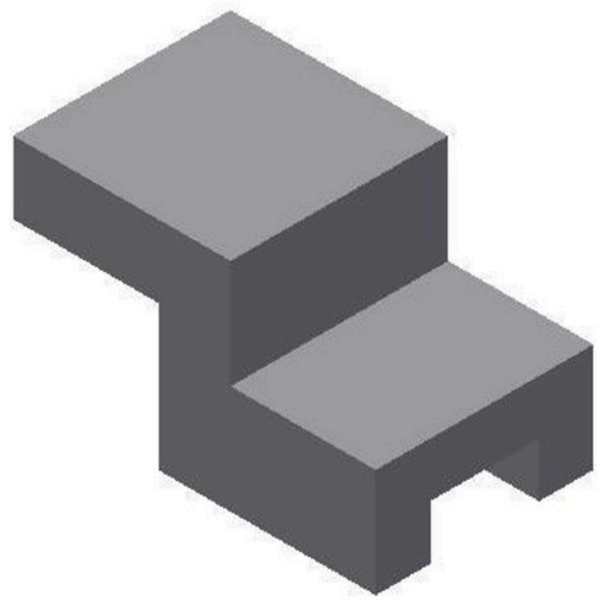


Figure Step 17

AUTHOR'S COMMENTS: In some computers, other background functions may have taken over the F keys. In that case, right-click somewhere in the Graphics window and then click on Hide All Constraints.

Step 18

Start a new sketch on the top plane as shown in the figure. (Figure Step 18)

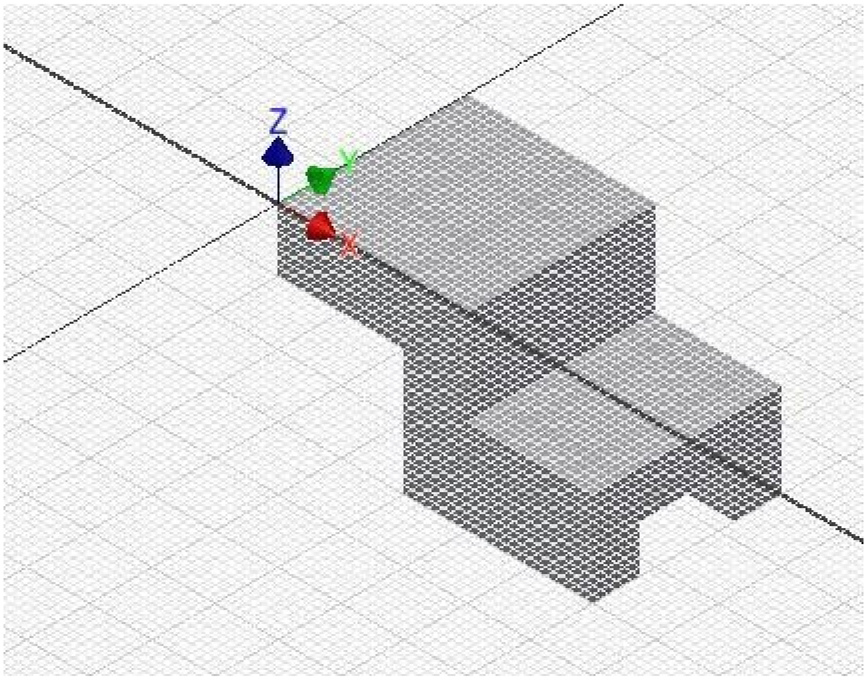


Figure Step 18

Step 19

Draw the 2D sketch on the new sketching plane and insert the necessary dimensions to fully constrain it. (Figure Step 19)

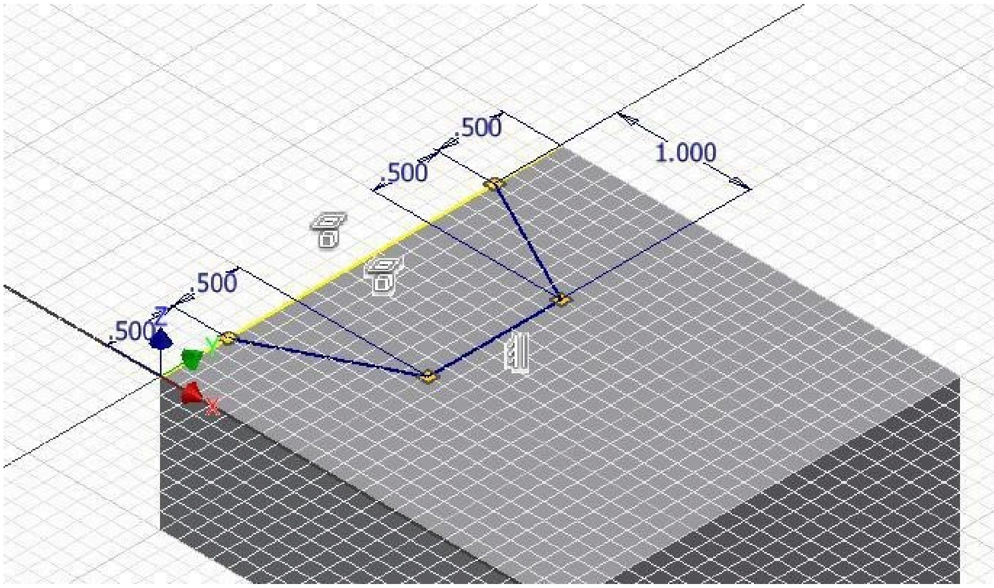


Figure Step 19 [Click to see image full size]

AUTHOR'S COMMENTS: Ensure that you snap onto the edge. That way, you will only be required to draw three lines to fully constrain the sketch.

Step 20

Extrude the top sketch to complete the solid model. (Figure Step 20)

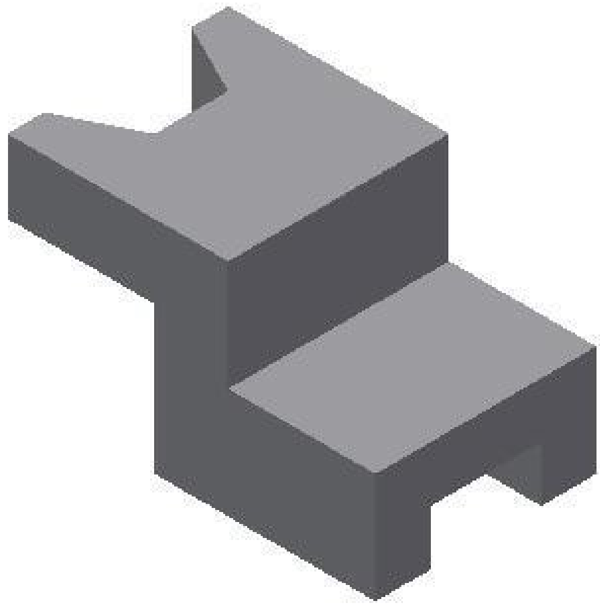


Figure Step 20

Step 21

Change the view to the Home view and apply the color: Chrome – Polished Black. Orbit the model to check the bottom. (Figure Step 21)

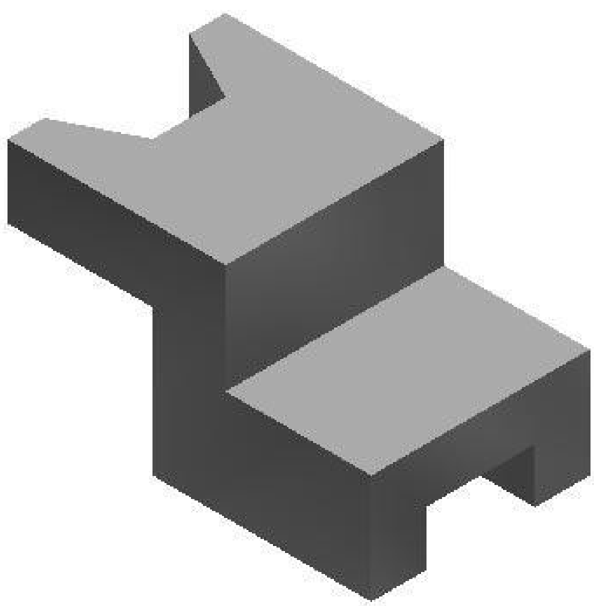


Figure Step 21

Step 22

Save and close the file.

Key Principles

Key Principles in Module 10

1. Inventor has three predefined planes that can be used to draw the Base sketch on. They are the XY, XZ and YZ planes. The XY plane is the Top view, the XZ plane is the Front view, and the YZ is the Right Side view of the Base model.
2. A consumed sketch is a 2D sketch that has been extruded or revolved. An unconsumed sketch is a 2D sketch that is blank or one that has not been extruded or revolved.

Lab Exercise 10-1

Time allowed: 45 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab Lab 10-1	Inventor Course	Inches	English-Modules Part (in).ipt	Titanium – Polished	N/A

Step 1

Project the Center Point onto the base sketching plane.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude them to produced the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. (Figure Step 2A and 2B)

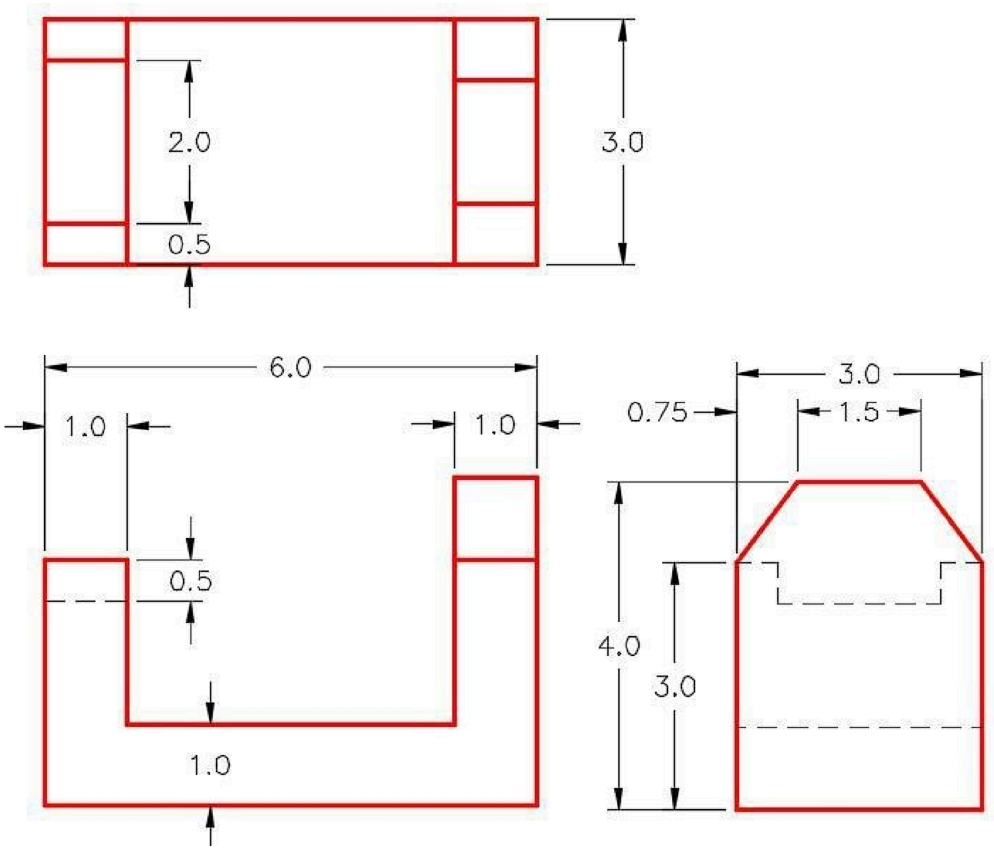


Figure Step 2A
Dimensioned Multiview Drawing [Click to see image full size]

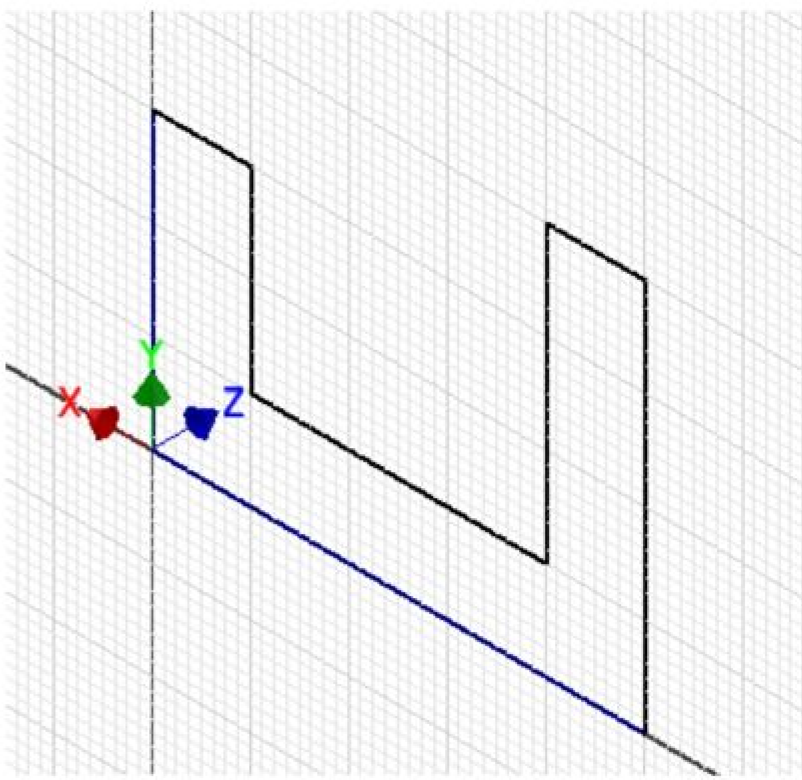


Figure Step 2B
Suggested Base Sketch – Front XZ Plane

Step 3

Apply the colour as shown above. (Figure Step 3)

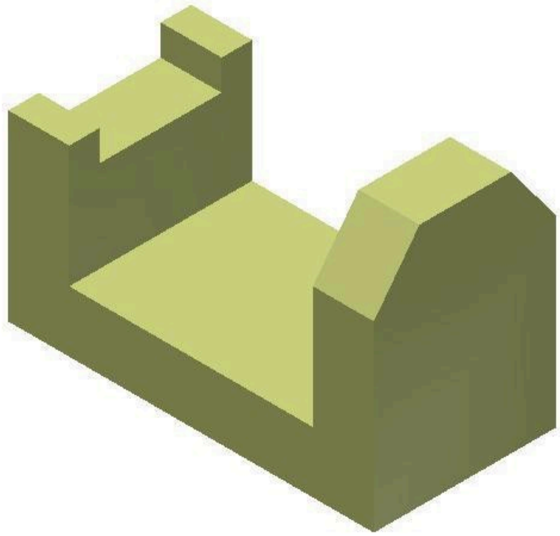
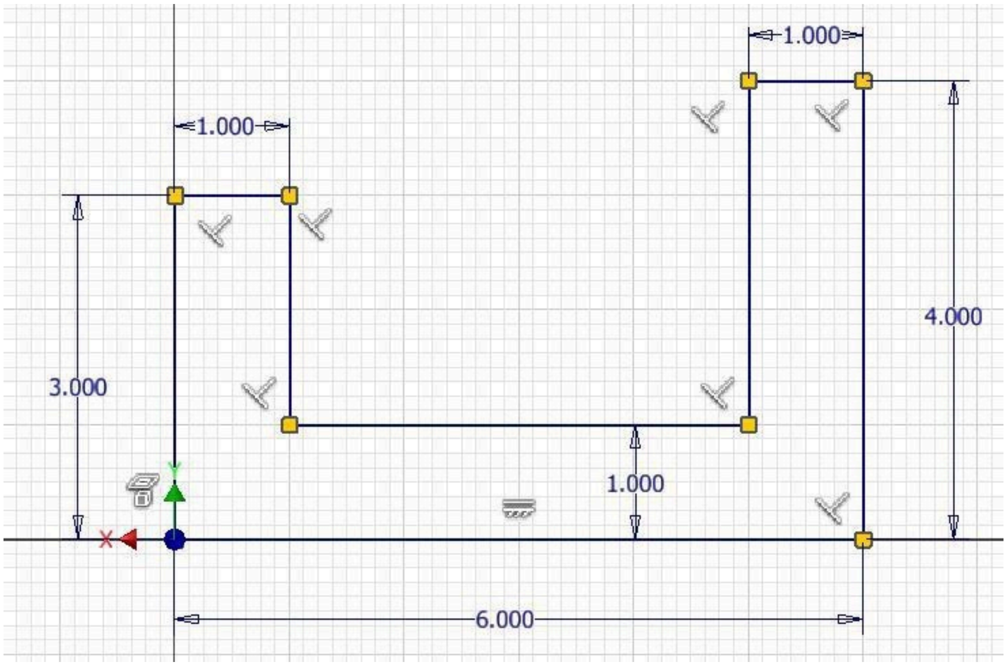
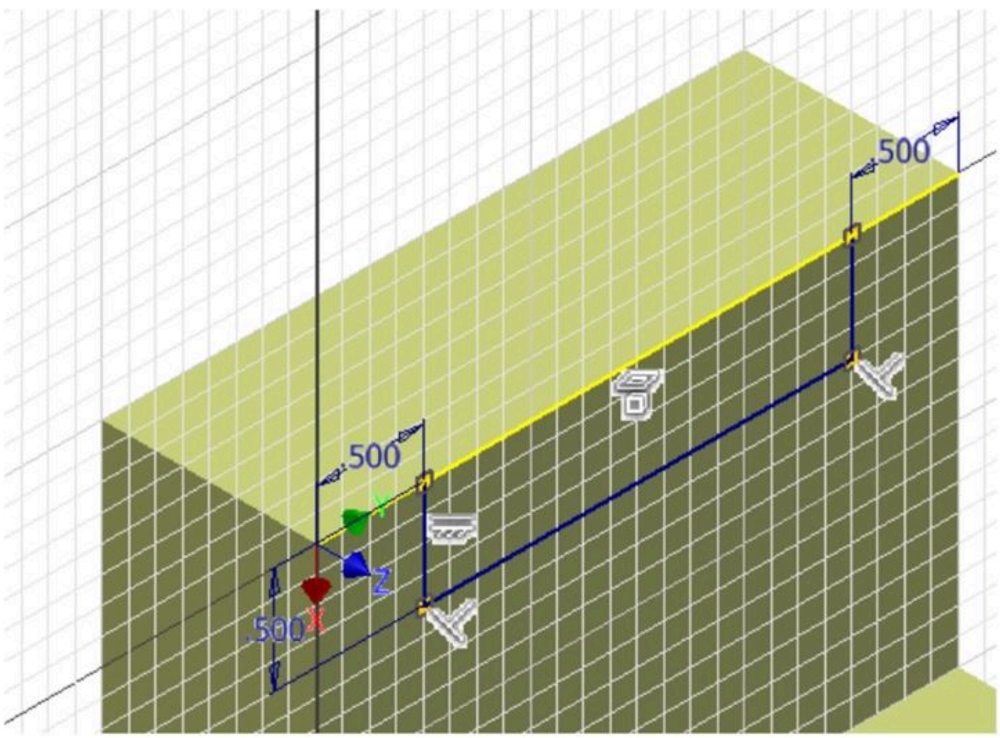


Figure Step 3 Completed Solid Model – Home View
[Click to see image full size]

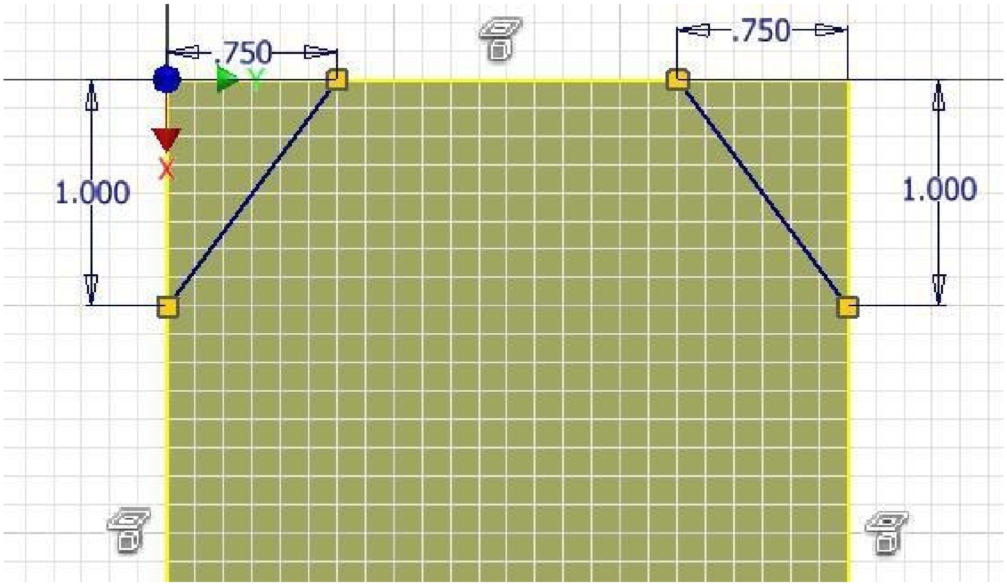
AUTHOR'S GEOMETRIC CONSTRAINS: The following three figures shows the base and additional sketch's construction method plus geometric and dimensional constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketches using different construction methods and constraints, that is what is important. You may want to compare your construction method and constraints used with the authors.



Author's Base Sketch [Click to see image full size]



Author's Sketch [Click to see image full size]



Author's Sketch [Click to see image full size]

AUTHOR'S COMMENTS: Your sketches may not match the figures exactly. Ensure that each sketch is fully constrained.

Lab Exercise 10-2

Time allowed: 45 minutes.

Part Name	Project	Units	Template	Color
Inventor Lab Lab 10-2	Inventor Course	Millimeters	Metric-Modules Part (mm).ipt	Zinc Chromate 2

Step 1

Project the Center Point onto the base sketching plane.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude them to produced the solid model. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. (Figure Step 2A and 2B)

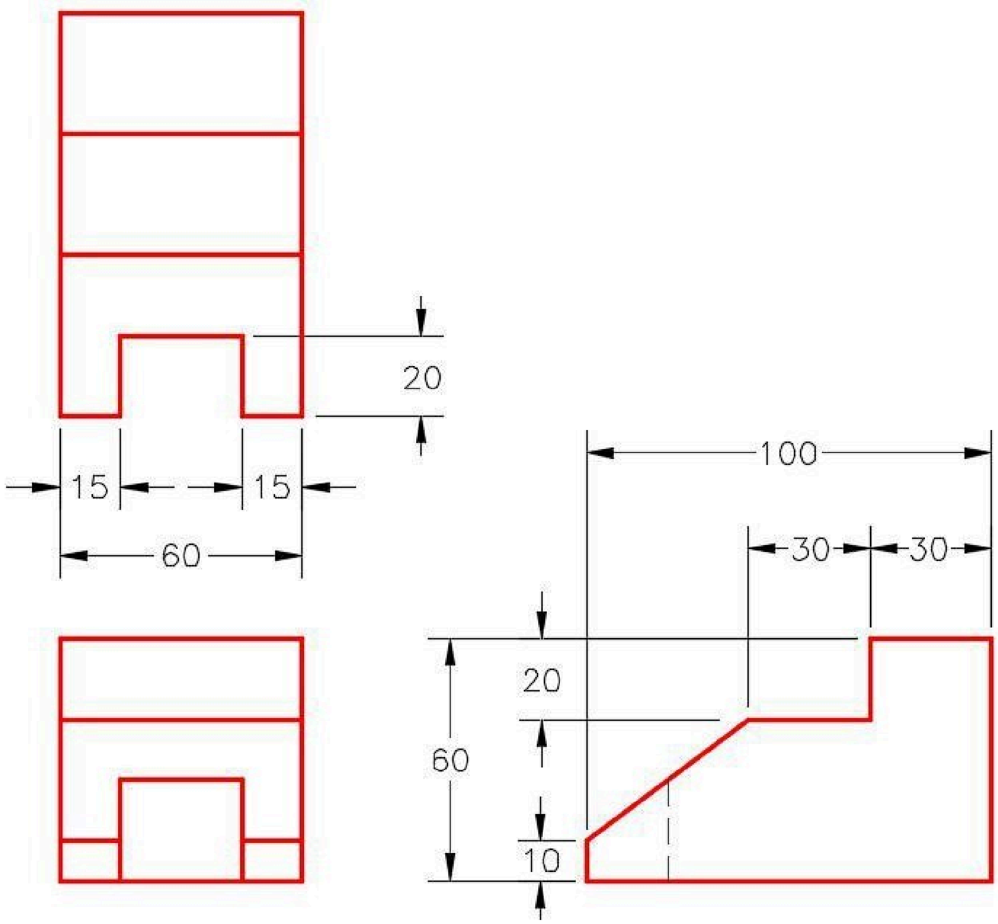


Figure Step 2A
Dimensioned Multiview Drawing [Click to see image full size]

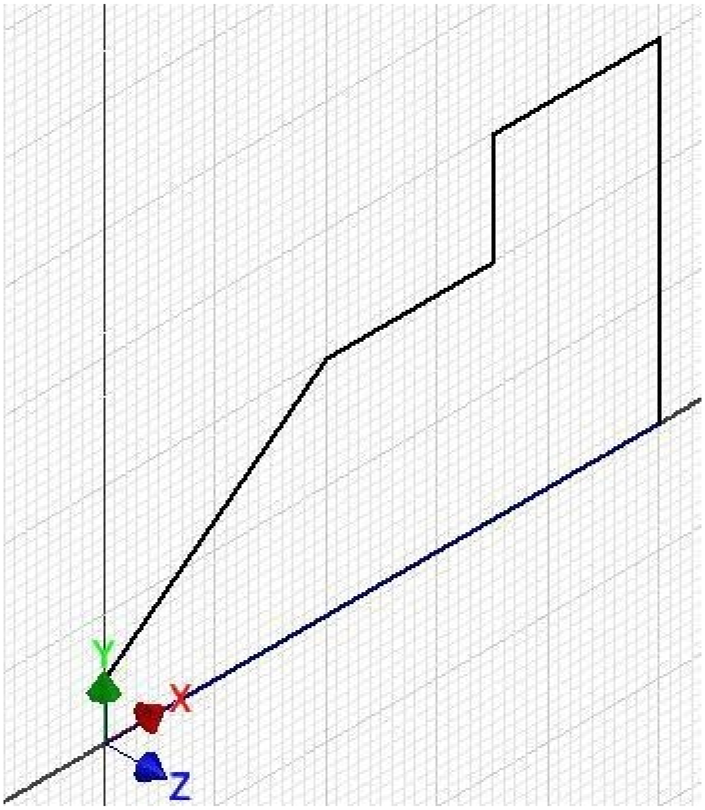


Figure Step 2B
Suggested Base Sketch – Right Side YZ Plane

Step 3

Apply the colour shown above. (Figure Step 3)

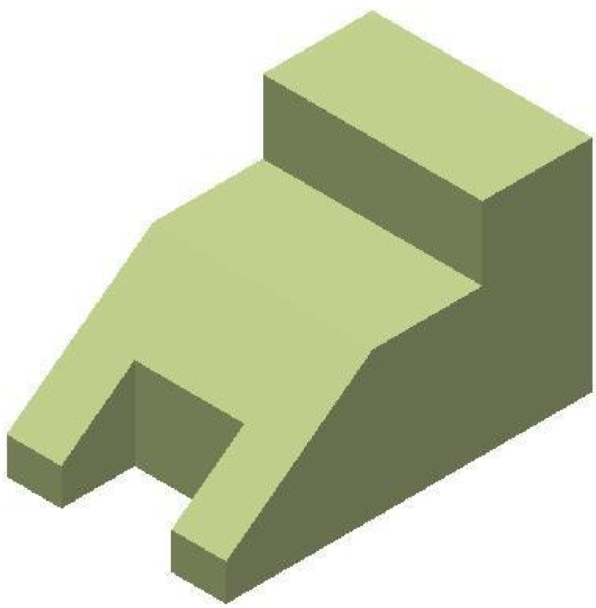
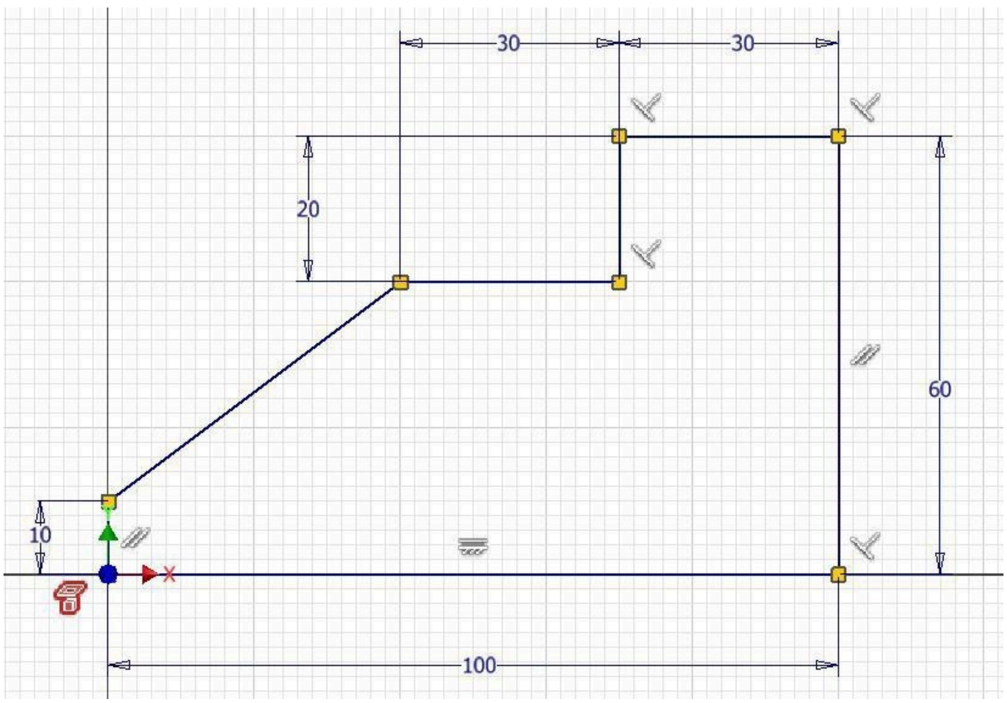
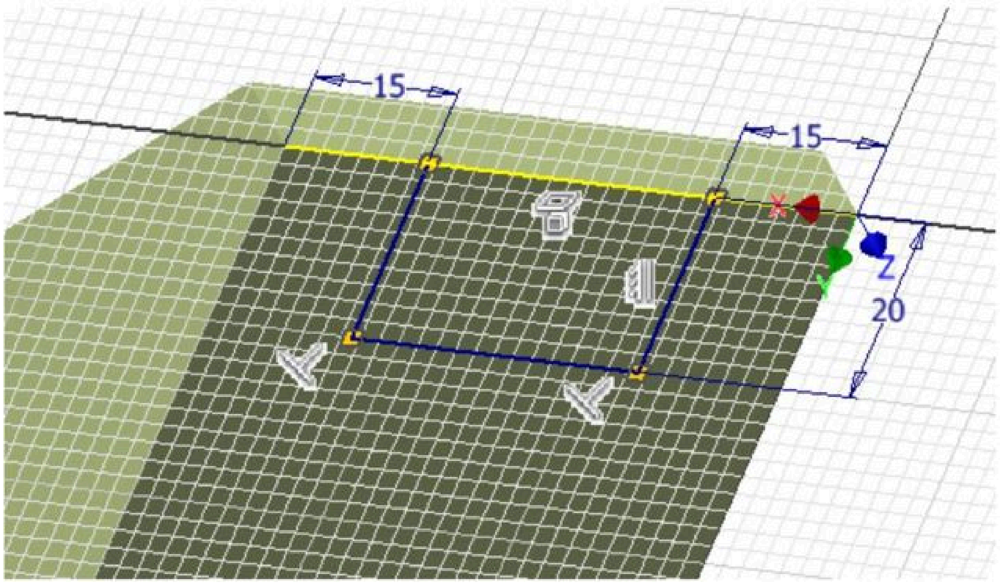


Figure Step 3 Completed Solid Model – Home View
[Click to see image full size]

AUTHOR'S GEOMETRIC CONSTRAINS: The following figures shows the sketch's construction method plus geometric and dimensional constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketch using a different construction method and constraints, that is what is important. You may want to compare your construction method and constraints used with the authors.



Author's Base Sketch [Click to see image full size]



Author's Sketch [Click to see image full size]

AUTHOR'S COMMENTS: Your sketches may not match the figures exactly. Ensure that each sketch is fully constrained.

Module 11 Competency Test No. 2 Open Book

Learning Outcomes

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

1. Within a one hour time limit, complete a written exam and the lab exercise.

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 you have completed this module.

Multiple Choice Questions

Select the BEST answer.

1. What view of the model is the XZ plane in Inventor?
   A. Left Side
   B. Right Side
   C. Top
   D. Bottom
   E. Front
2. Which one of Inventor's menus shows if a sketch is consumed or unconsumed?
   A. Status bar
   B. Standard menu
   C. Browser bar
   D. Panel bar
   E. 2D Sketch panel
3. What command is used to draw a 2D Sketch on the existing solid model?
   A. NEW
   B. LOOK AT

C. 2D SKETCH

D. EXTRUDE

E. LINE

4. Which plane is the default plane used in the templates supplied with the Inventor book?

A. ZY

B. XZ

C. XY

D. YX

E. YZ

5. Which of the following best describes a consumed 2D sketch?

A. It is blank.

B. It has been extruded or revolved to create a 3D feature.

C. It has been deleted.

D. It cannot be edited.

E. It has been dimensioned.

6. In Figure 11-1, what is the purpose of the middle icon in the EXTRUDE dialogue box?

A. Cut

B. Subtract

C. Add

D. Join

E. Intersect

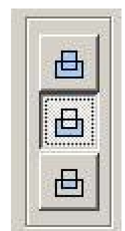


Figure 11-1

7. What kind of dimension does a linear dimension insert?

A. Line

B. Object

C. Horizontal only

D. Vertical only

E. Delta X or Delta Y

8. What view of the model is the XY plane in Inventor?

A. Left Side

B. Right Side

C. Top

D. Bottom

E. Front

9. On the colour scheme High Contrast, what colour will the sketch appear when it is fully constrained?
    A. Black
    B. White
    C. Purple
    D. Blue
    E. Green
10. What view of the model is the YZ plane in Inventor?
    A. Rotated
    B. Right Side
    C. Top
    D. Bottom
    E. Front

Lab Exercise 11-1 Part B

Time allowed: 60 minutes.

Part Name	**Project**	**Units**	**Template**	**Color**	**Material**
Inventor Lab Lab 11-1	Inventor Course	Millimeters	Metric-Modules Part (mm).ipt	Orange	N/A

Step 1

Project the Center Point onto the Base sketch.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude them to produced the solid model. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. Ensure that all lines on all sketches display purple on a black background. (Figure Step 2A and 2B)

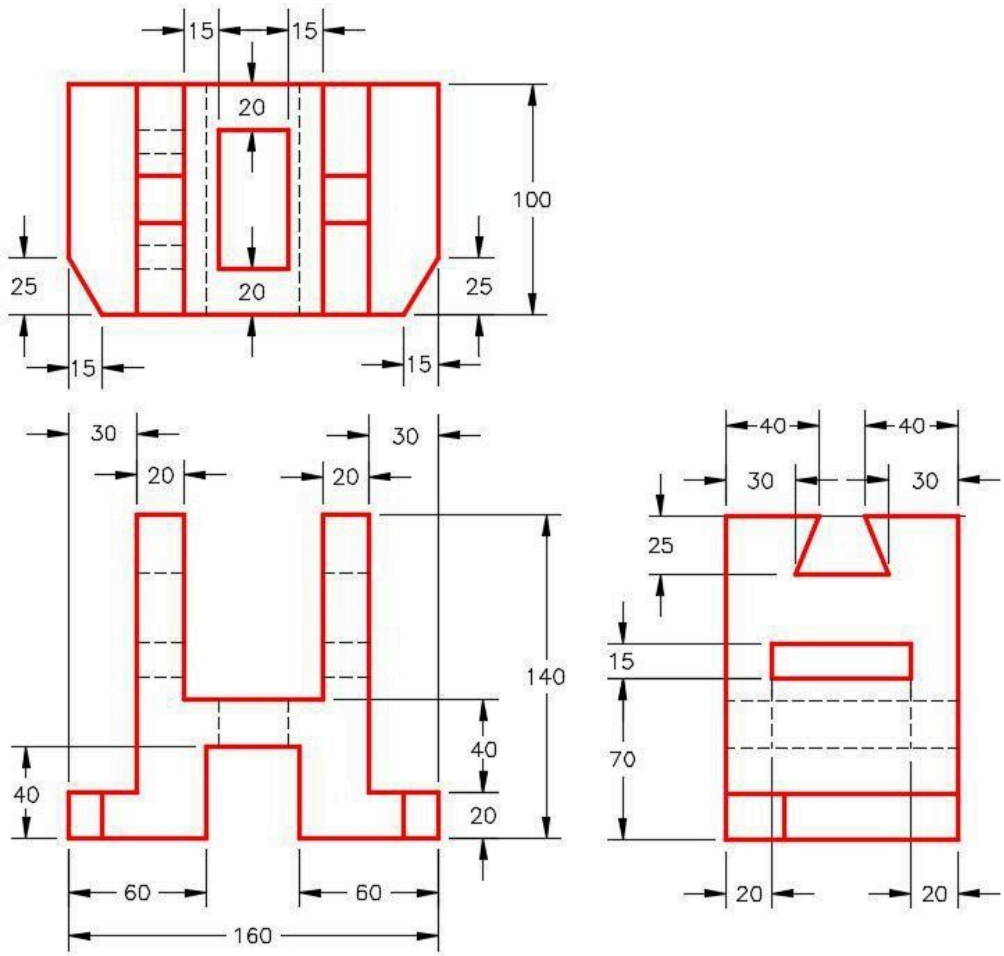


Figure Step 2A
Dimensioned Drawing [Click to see image full size]

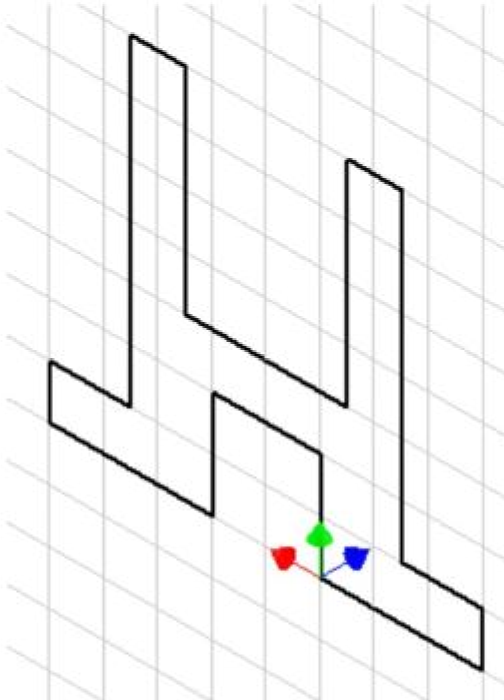


Figure Step 2B
Suggested Base View –
Front – XZ Plane

Step 3

Apply the colour shown above. (Figure Step 3A, 3B, and 3C)

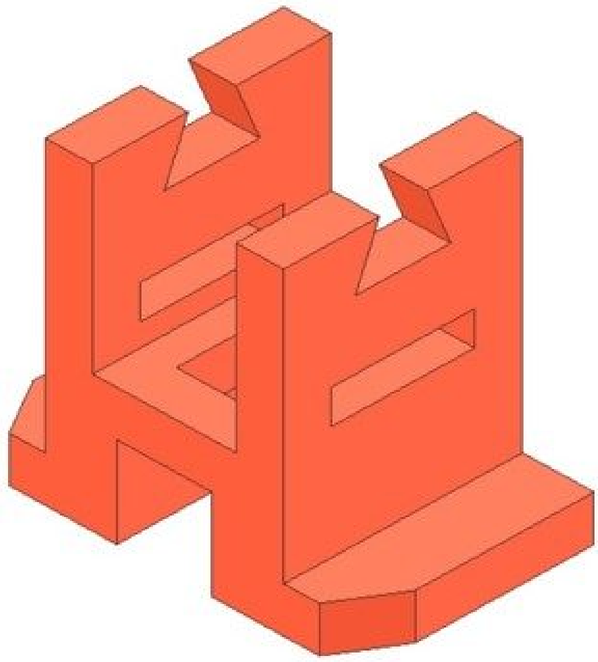


Figure Step 3A
Completed Solid Model –
Home View

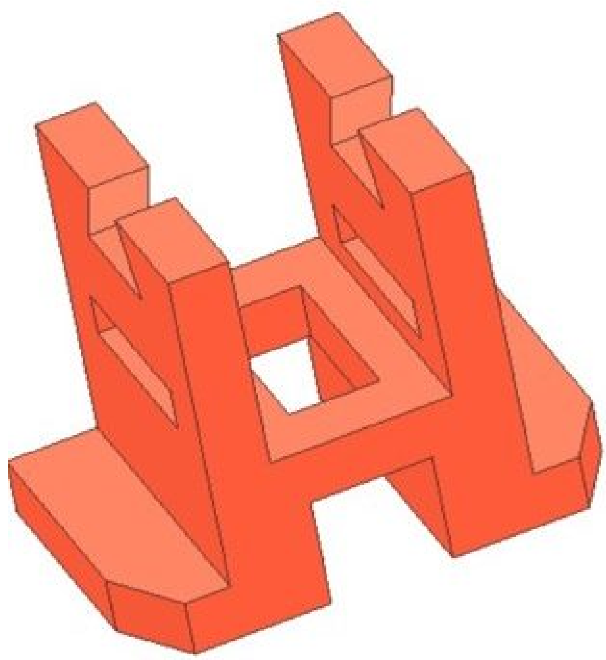


Figure Step 3A
Solid Model – Rotated View

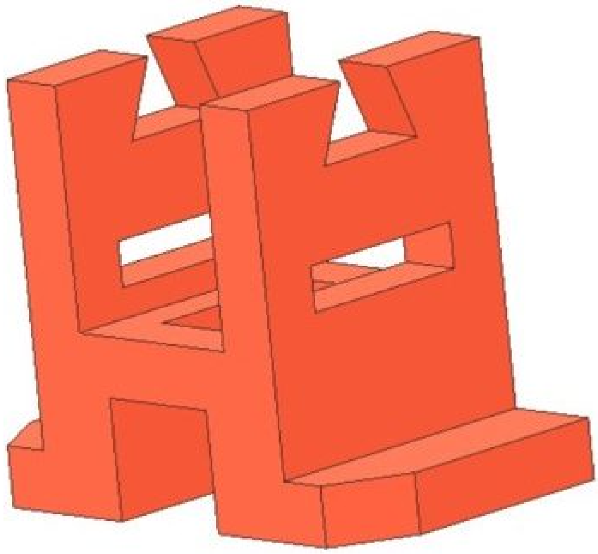


Figure Step 3A
Solid Model – Rotated View

Part 3

Module 12 Circles

Learning Outcomes

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

1. Describe construction objects and their use in a 2D sketch.
2. Describe drawing circles and offsets in a 2D sketch.
3. Apply the CENTER POINT CIRCLE and OFFSET commands to complete 2D sketches.

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

Study the drawings in Figure 12-1, 12-2, 12-3, and 12-4 for a description of the geometry of a circle.

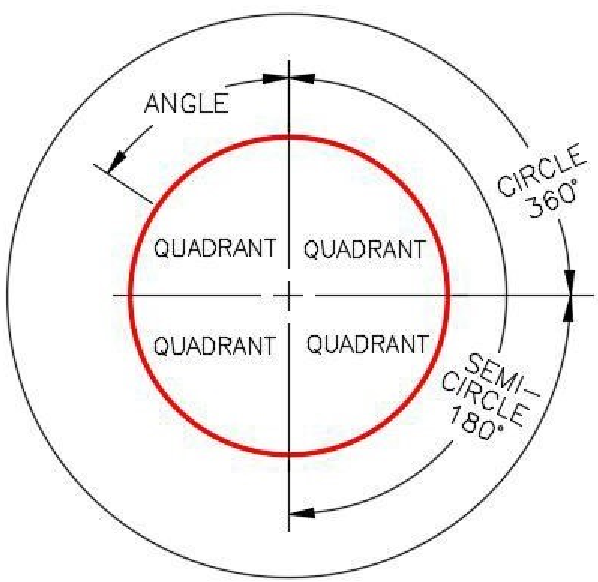


Figure 12-1
Geometry of a Circle – Part 1 [Click to see image full size]

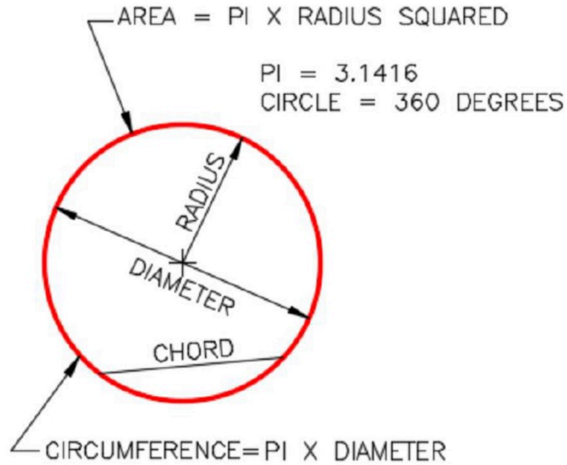


Figure 12-2
Geometry of a Circle – Part 2 [Click to see image full size]

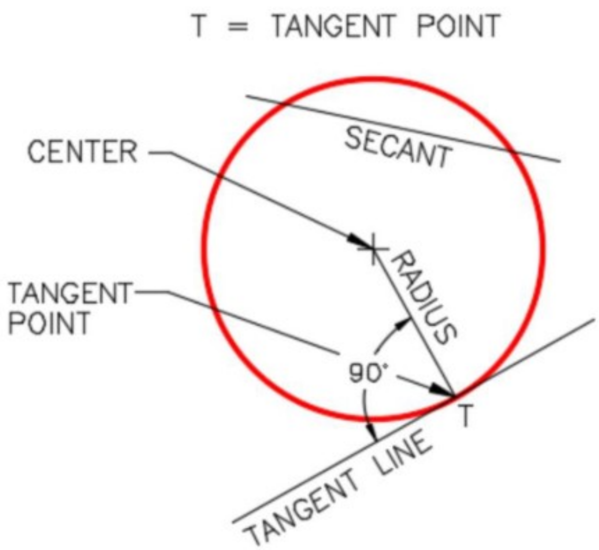


Figure 12-3
Geometry of a Circle – Part 3 [Click to see image full size]

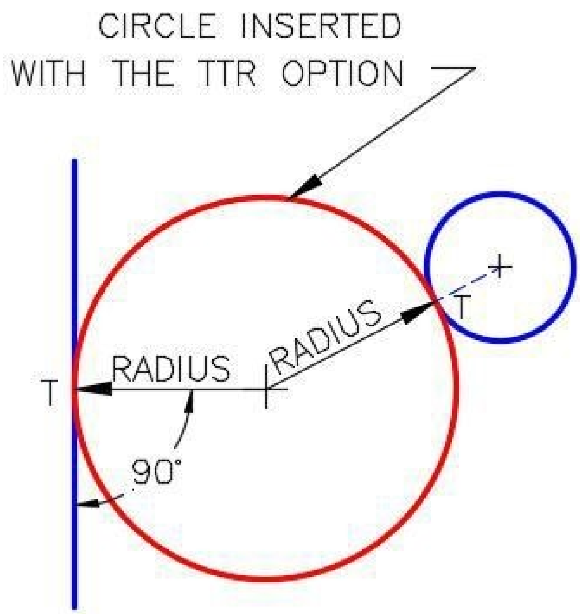


Figure 12-4
Geometry of a Circle – Part 4

Drafting Lesson: Reading Dimensions for Circles and Arcs

When reading the dimensions for circles and arcs, consider the following: Circles are dimensioned as diameters. For example: 2.0 DIA.

Arcs are dimensioned as radii. For example: 1.5 R

When there is more than one circle of the same diameter, they are only dimensioned once. For example: 0.5 DIA., 4 PLACES

Sometimes multiple arcs are dimensioned as typical (TYP.). For example: 2.0 R TYP. This simply means that there is at least one additional arc of the same size, See Figure 12-5.

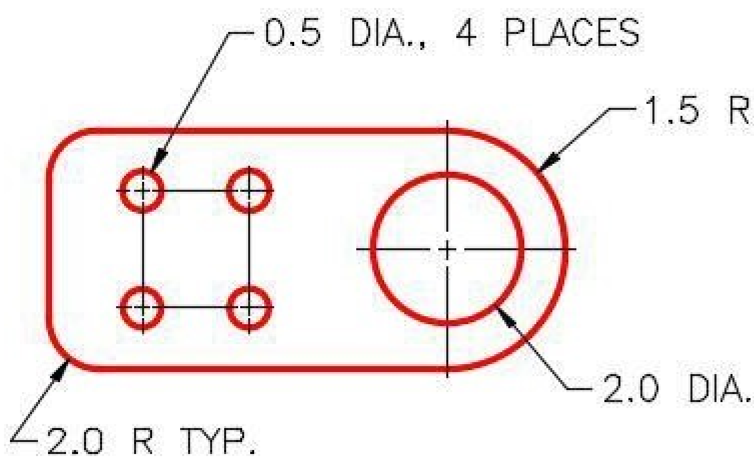
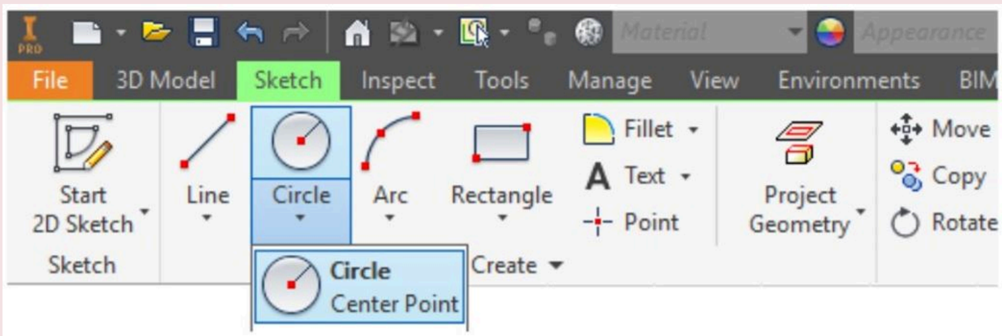


Figure 12-5
Typical Drawing

Inventor Command: CENTER POINT CIRCLE

The CENTER POINT CIRCLE command is used to draw a circle by entering the location of its centre point and its radius.

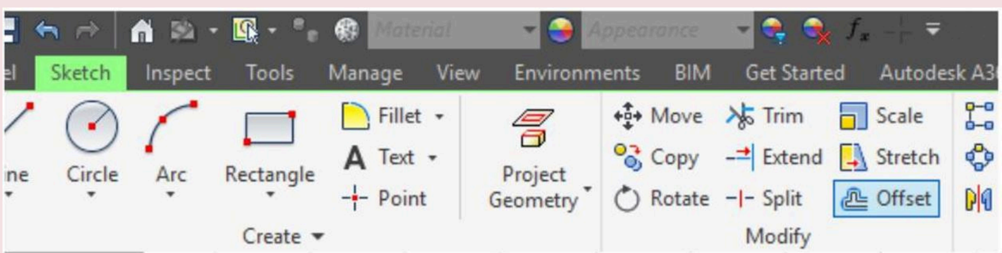
Shortcut: **C**



Inventor Command: OFFSET

The OFFSET command is used to draw an object parallel to an existing object.

Shortcut: **O**



Construction objects are objects that are drawn in the sketch to assist you to complete the sketch but will be ignored by Inventor when the sketch is extruded or revolved to create the Base model. Geometrical and dimensional constraints can be applied to construction objects the same as they are to drawing objects. Construction objects and drawing objects only differ in the properties of the objects. They are both drawn and manipulated the same way in a 2D sketch.

There are two methods of creating construction objects as follows:

Method 1

Enable the Construction icon, see Figure 12-6, and then draw the objects. All objects drawn while the Construction icon is enabled, will be created as construction objects.

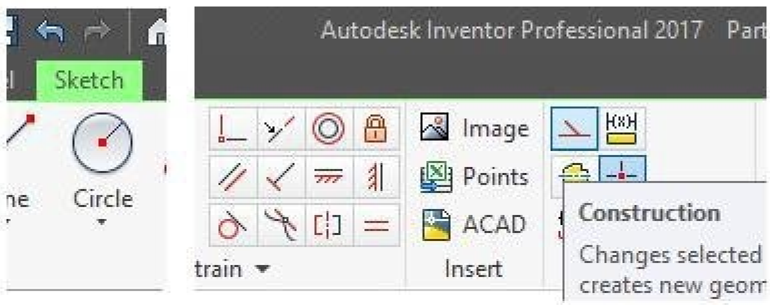


Figure 12-6

Method 2

Draw the object with the Construction icon disabled. After the drawing object(s) is created, select it. While it is selected, click the Construction icon. Construction objects will appear as a dashed lines as shown in Figure 12-7.

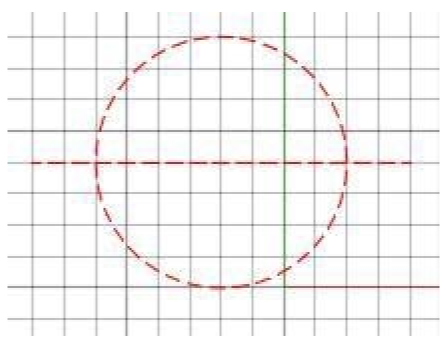
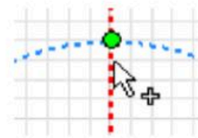


Figure 12-7
Construction Objects

	Geometrical Constraints		
Constraint	**Symbol**	**Icon**	**Definition**
Equal	=		An equal constraint resizes objects to be equal in size to a selected existing object.

Snap Symbols		
Mode	**Icon**	**How they appear in Inventor**
Intersection (Snaps to the intersection of two objects)		

USER TIP: Some commands require you to select one or more drawing objects when there is no command active. For example, the DELETE command. The Status bar will display Ready when there is no command active. To select one object at a time, select the object by clicking it with the left mouse button. The selected object will highlight and change colour. To select more then one object at time, hold down either the CTRL or SHIFT key while selecting the objects. When all objects have been selected, click the command icon or enter the command on the command line.

WORK ALONG: Drawing Circles and Cylindrical Models

Step 1

Check the default project and if necessary, set it to Inventor Course.

Step 2

Enter the NEW command to start a new part file using the template: English-Modules Part (in).ipt.

Step 3

Save the file with the name: Inventor Workalong 12-1. (Figure Step 3A and 3B)

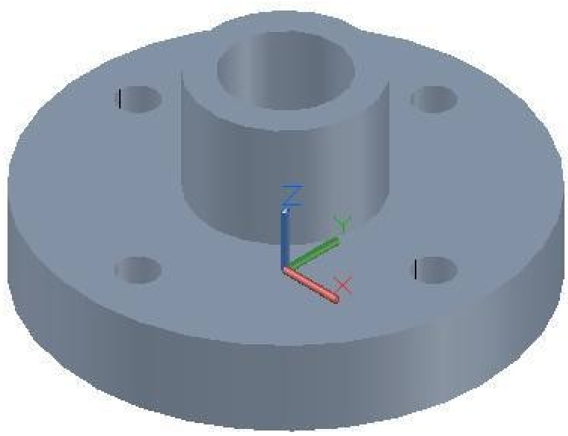


Figure Step 3A
3D Model
Home View [Click to see image full size]

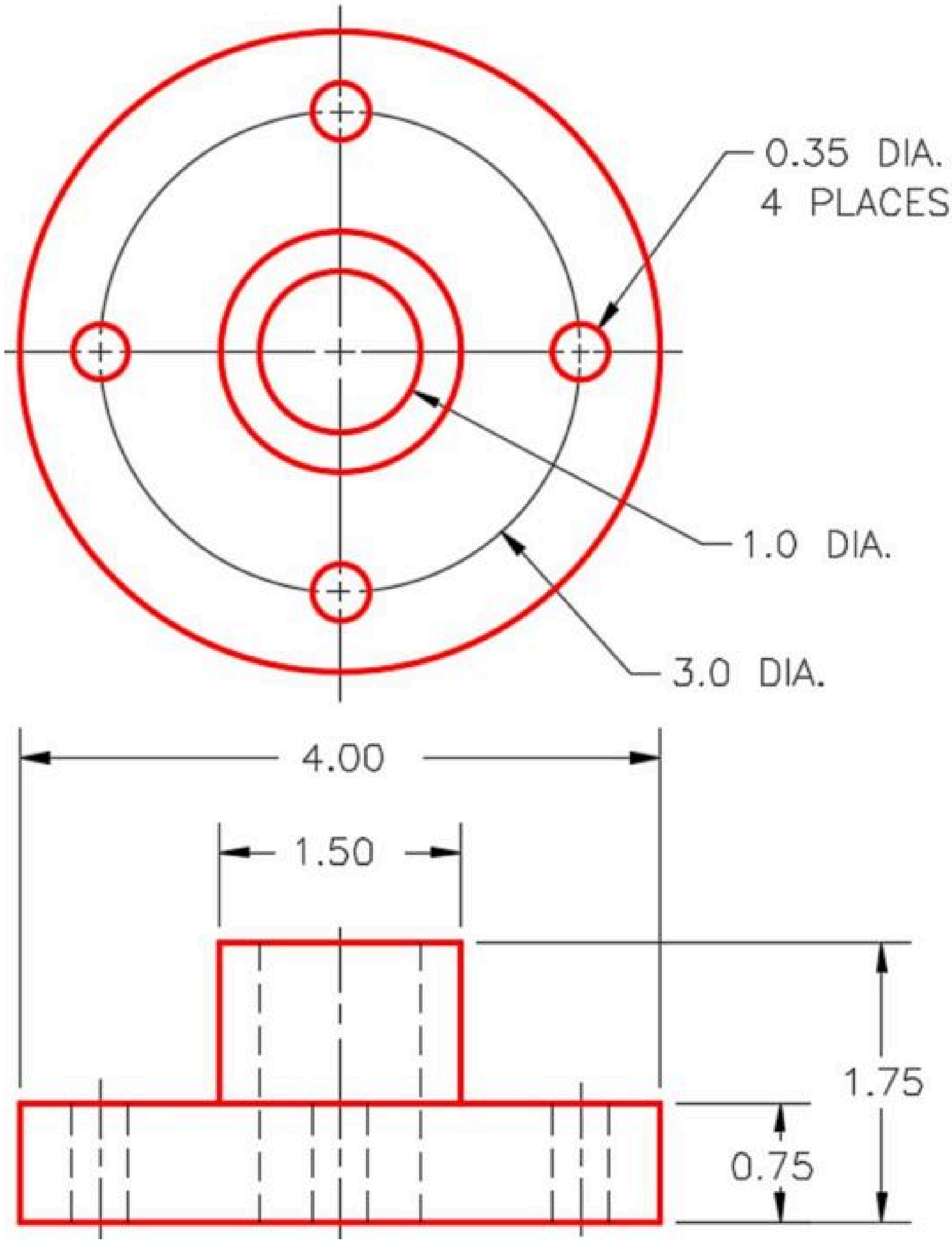


Figure Step 3B
Dimensioned Multiview Drawing [Click to see image full size]

Step 4

Draw the Base sketch on the Top view. Since this is the XY plane (the default plane), use Sketch1.

Step 5

Project the Center Point onto the sketching plane.

Step 6

Enter the CENTER POINT CIRCLE command and draw a 4.0 inch diameter circle. Watch the Status bar. The first prompt will be to select the centre point for the circle. For the centre point location, snap to the Center Point that was projected onto the sketching plane. Select the radius by guessing 2 inches from the centre. (Figure Step 6)

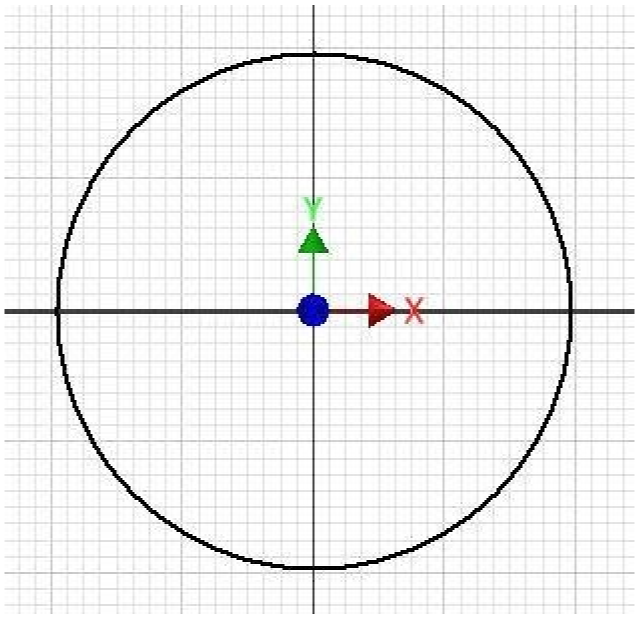


Figure Step 6

AUTHOR'S COMMENTS: You can guess at the location of the radius since you will be constraining its size with a dimension later. Ensure that the green dot appears when you snap to the Center Point.

Step 7

Click the OFFSET command and when prompted, select the circle. Right-click the mouse. In the Right-click menu, ensure that Loop Select is enabled. Move the mouse towards the centre, guess at the diameter size of the 1.0 inch. (Figure Step 7A, 7B, and 7C)

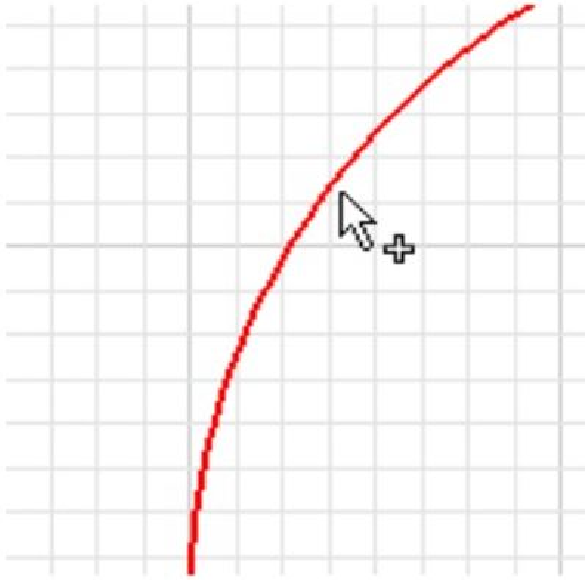


Figure Step 7A

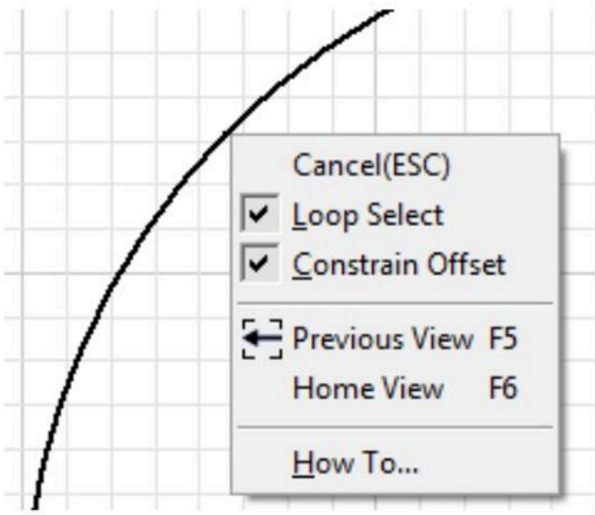


Figure Step 7B

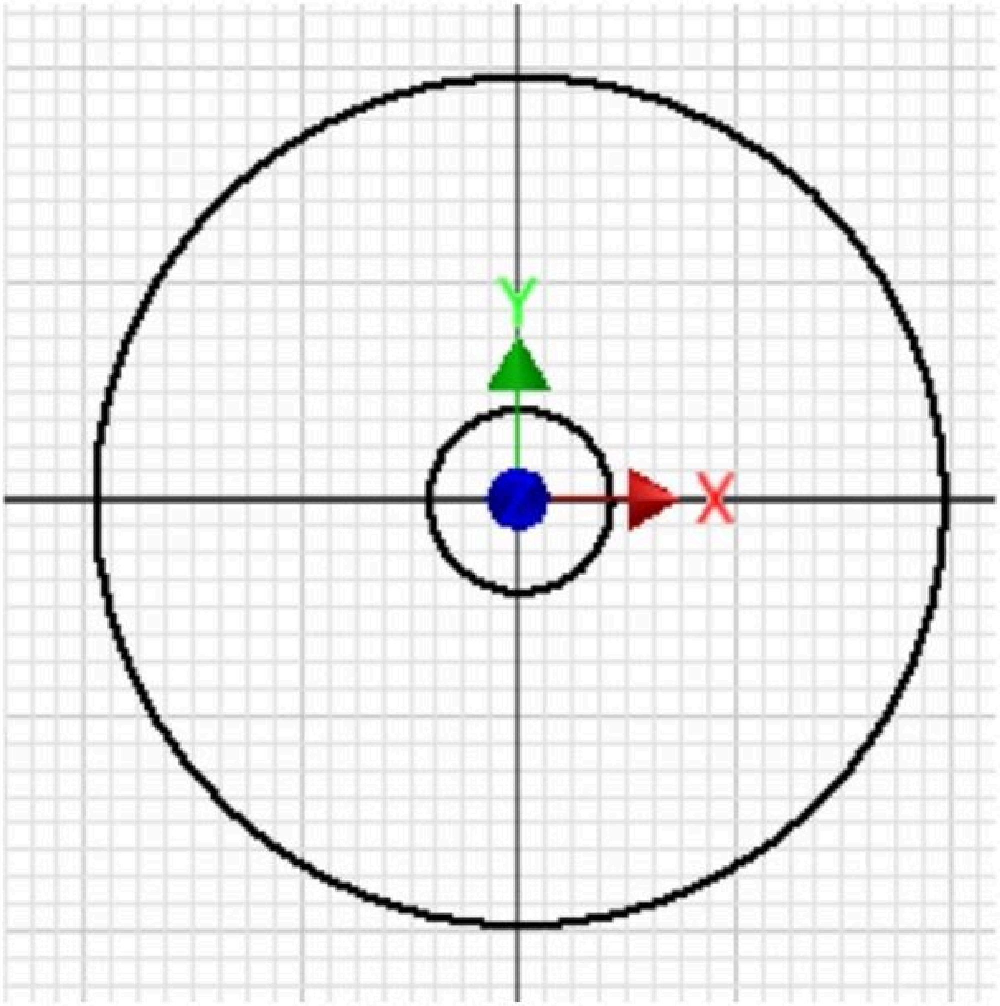


Figure Step 7C [Click to see image full size]

AUTHOR'S COMMENTS: Using the OFFSET command to draw the two additional circles will automatically constrain their centre to the centre of the offset circle.

Step 8

Click the Construction icon to enable it. (Figure Step 8)

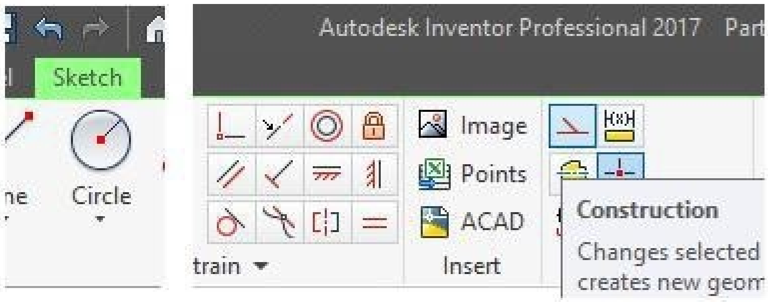


Figure Step 8

Step 9

Using what you just learned, enter the OFFSET command and offset the large circle to draw the 3 inch diameter construction circle. (Figure Step 9)

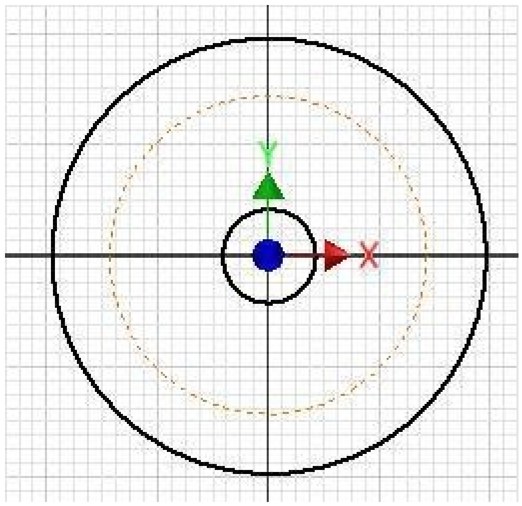


Figure Step 9

AUTHOR'S COMMENTS: By inserting a construction circle, it will not be part of the extrusion later. Guess at the radius as you will be constraining its size with a dimension later.

Step 10

Click the Tools menu. Click the Applications Options icon to open the Applications Options dialogue box. Enable the Sketch tab. In the Display area, disable Axes. Click OK to close the dialogue box. (Figure Step 10A and 10B)

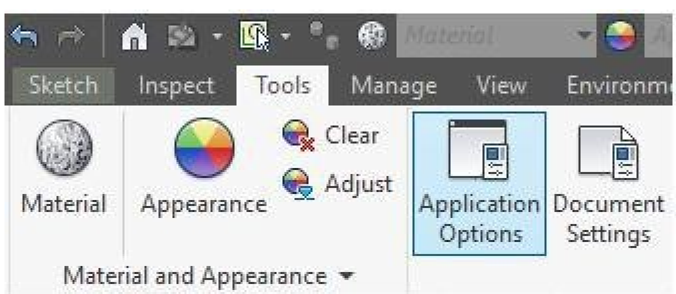


Figure Step 10A

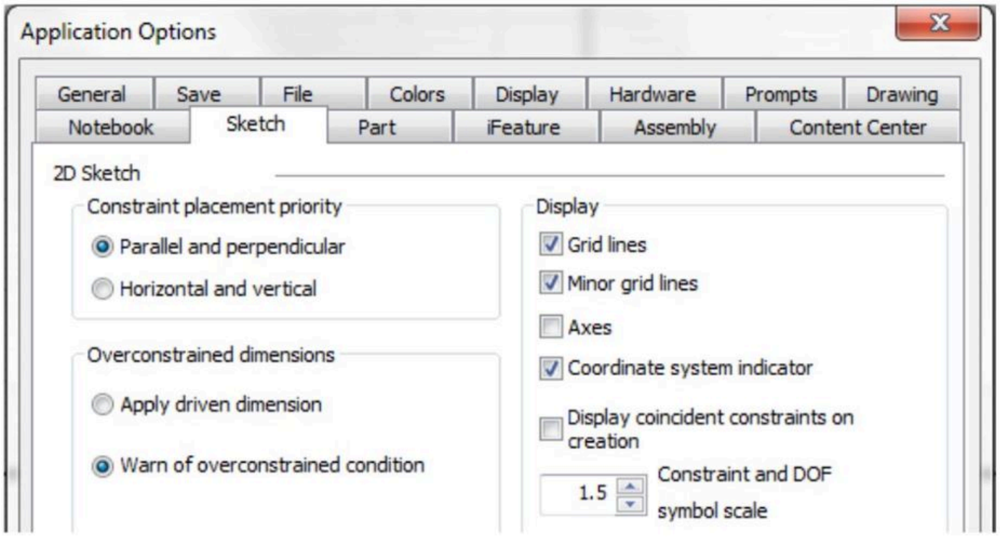


Figure Step 10B [Click to see image full size]

AUTHOR'S COMMENTS: I am having you disable the display of the axes as they get in the way of drawing this sketch. If you wish, you can leave them disabled or enable their display after you complete this workalong.

Step 11

Dimension the three circles. Press F8 to display the geometrical constraint icons. (Figure Step 11)

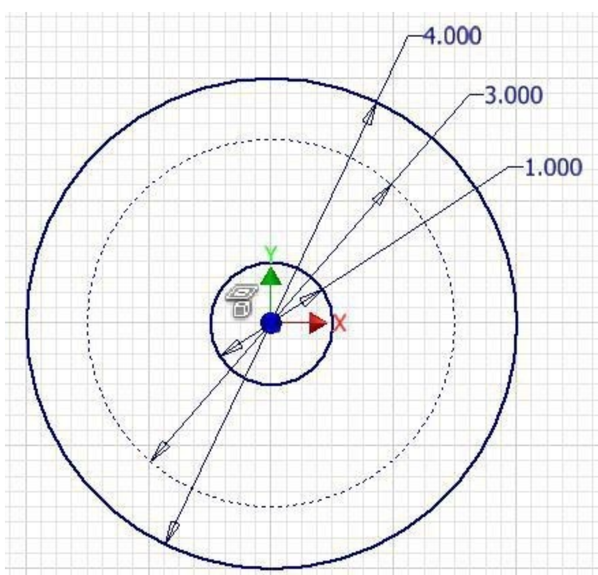


Figure Step 11 [Click to see image full size]

AUTHOR'S COMMENTS: Your sketch should be fully constrained.

Step 12

Using the LINE command, draw a line by first snapping to the Center Point and ending it by snapping onto the circle. Make sure that the Horizontal constraint icon displays when you are drawing the line. The constraint icons should appear similar to the figure. (Figure Step 12)

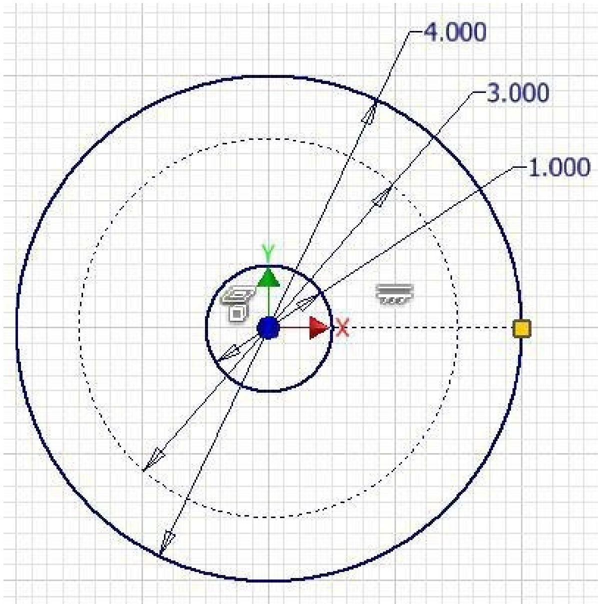


Figure Step 12 [Click to see image full size]

> **AUTHOR'S COMMENTS:** Since the Construction Line icon is enabled from Step 8, this will be a construction line. In other words, used to construct and constrain the sketch but not included in the extruded.

> **AUTHOR'S COMMENTS:** Ensure that you snap to the Center Point and onto the circle circumference for the other end.

Step 13

Using the LINE command, draw three additional construction lines by first snapping to the Center Point and ending them by snapping onto the circle. The constraint icons should appear similar to the figure. (Figure Step 13)

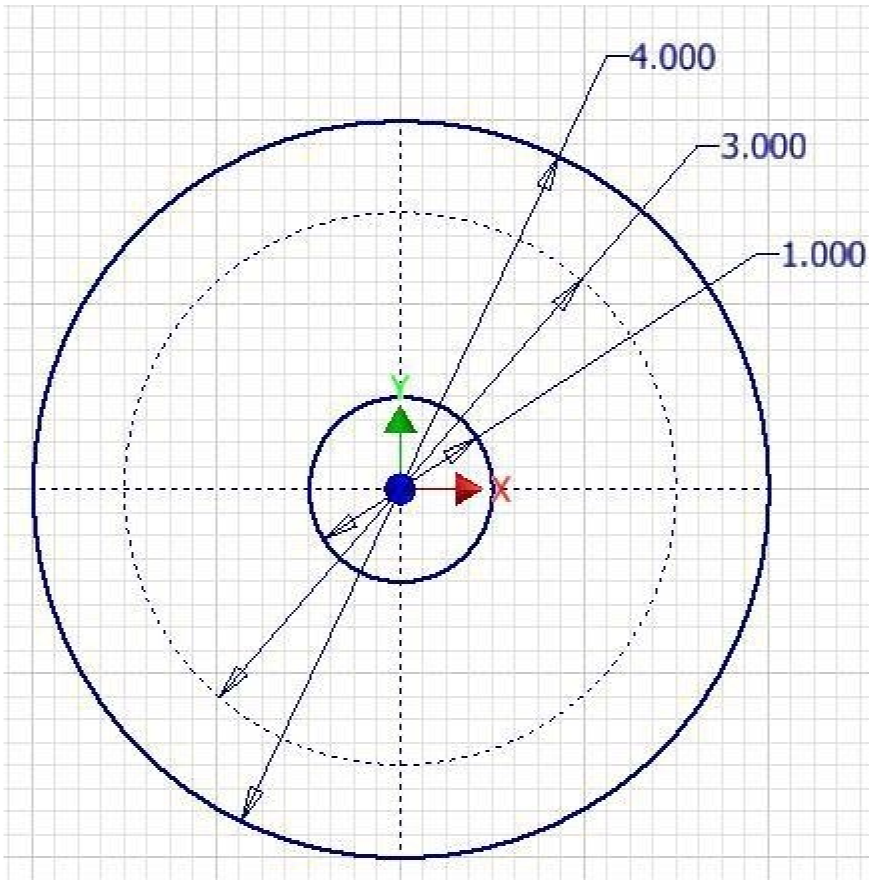


Figure Step 13

AUTHOR'S COMMENTS: Your sketch should be fully constrained.

Step 14

Press F8 to display the constraint icons. (Figure Step 14)

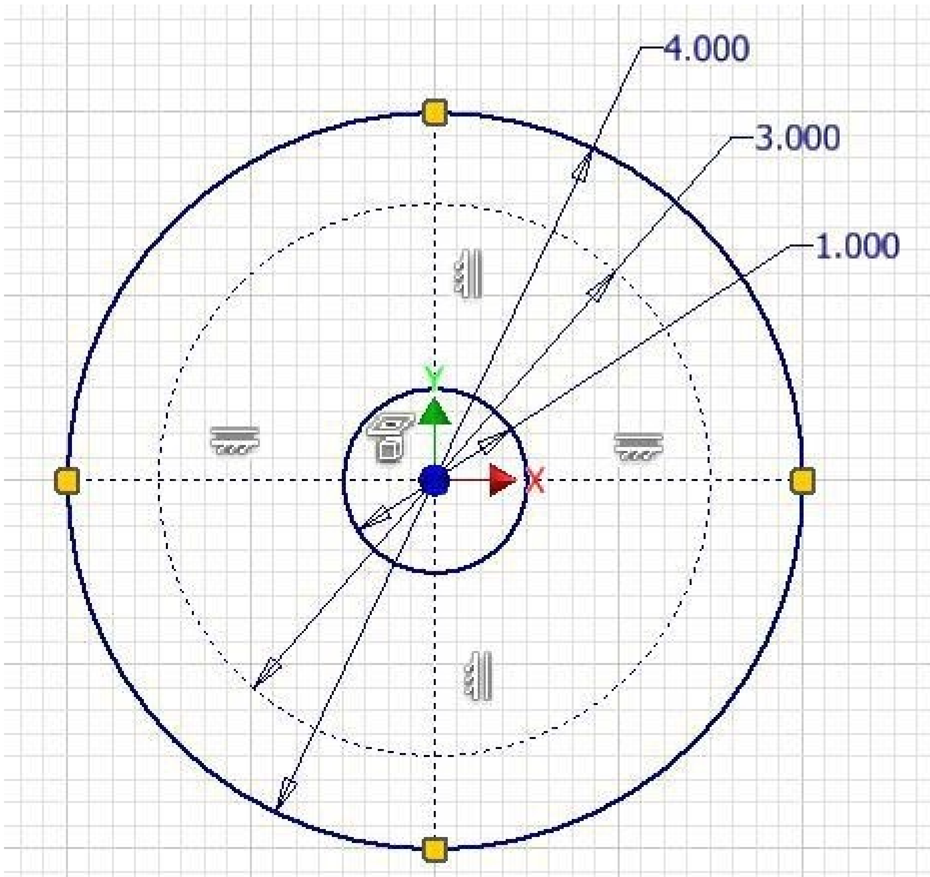


Figure Step 14

Step 15

Click the Construction icon to disable it.

Step 16

Enter the CENTER POINT CIRCLE command. With the cursor in the sketching window, right-click the mouse. In the Right-click menu, select Intersection. Select the construction circle and then the construction line. Notice how the cursor will display a plus sign beside it. When you select the line, the green snap indicator will display. Select a location for the radius of the circle by guessing at its size. (Figure Step 16A, 16B, 16C, and 16D)

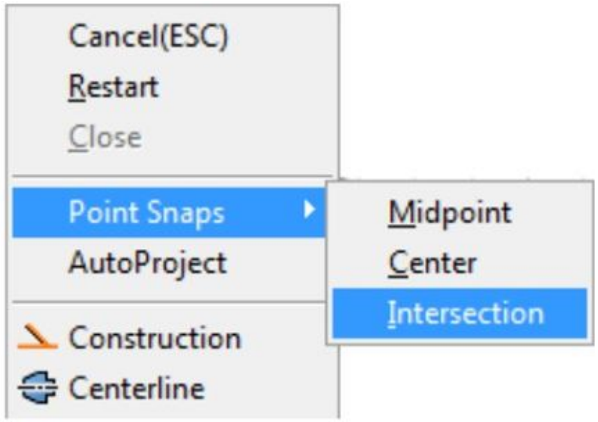


Figure Step 16A

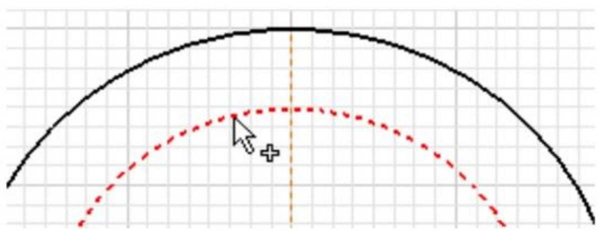


Figure Step 16B [Click to see image full size]

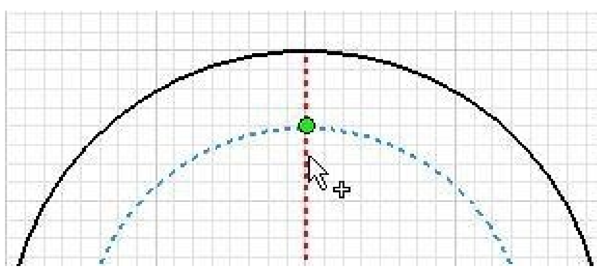


Figure Step 16C [Click to see image full size]

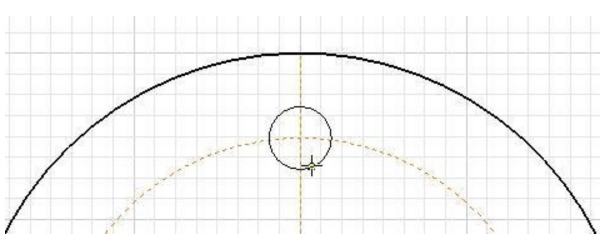


Figure Step 16D [Click to see image full size]

Step 17

Using what you just learned in Step 14, insert the 3 additional circles. For now, the diameter of the circles is not important. (Figure Step 17)

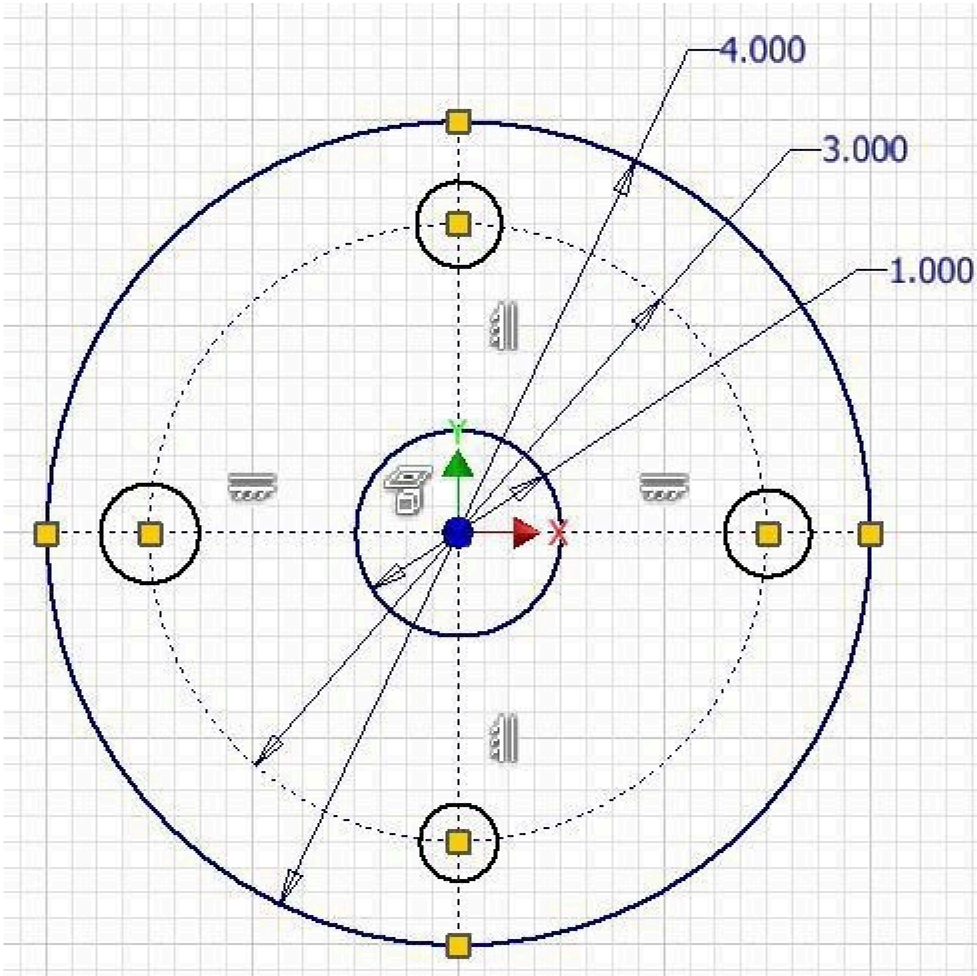


Figure Step 17 [Click to see image full size]

AUTHOR'S COMMENTS: Ensure that you snap to the intersection of the construction circle and the construction line when you locate the centre for each circle.

Step 18

Dimension one circle only. (Figure Step 18)

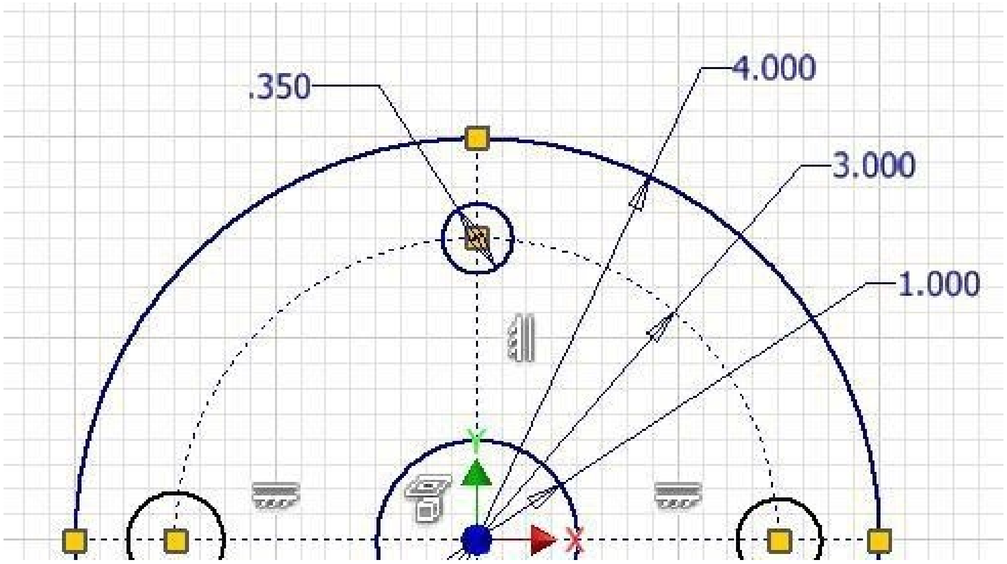


Figure Step 18 [Click to see image full size]

Step 19

Ensure that there is no active command and select the Equal constrain icon. (Figure Step 19)

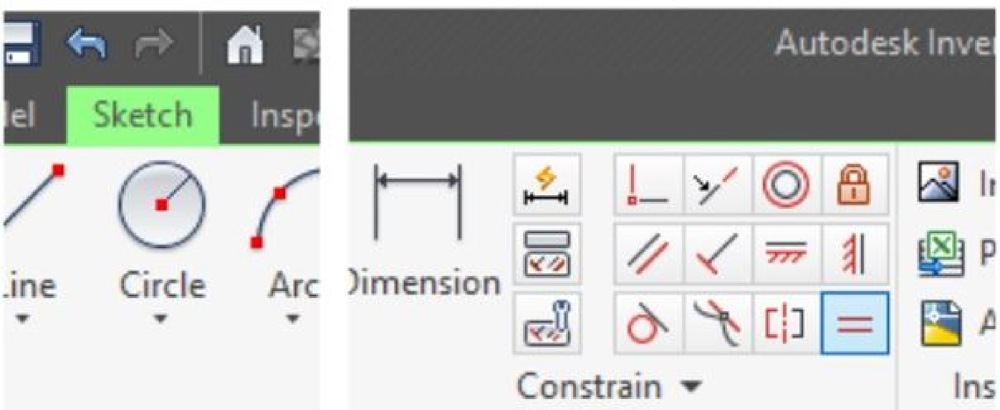


Figure Step 19

Step 20

For the first circle, select the top (dimensioned circle) and for the second circle select one of the other circles. It will constrain the second circle as equal in size to the dimensioned circle. (Figure Step 20A and 20B)

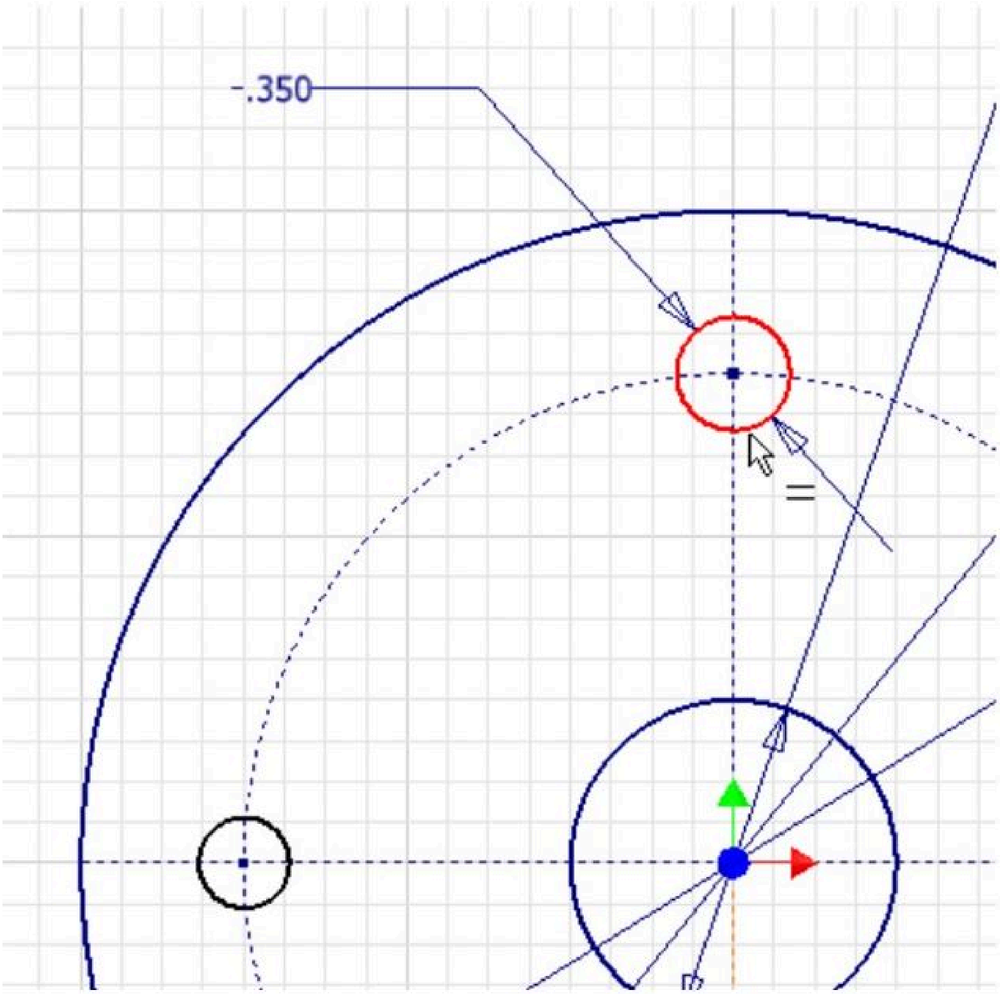


Figure Step 20A [Click to see image full size]

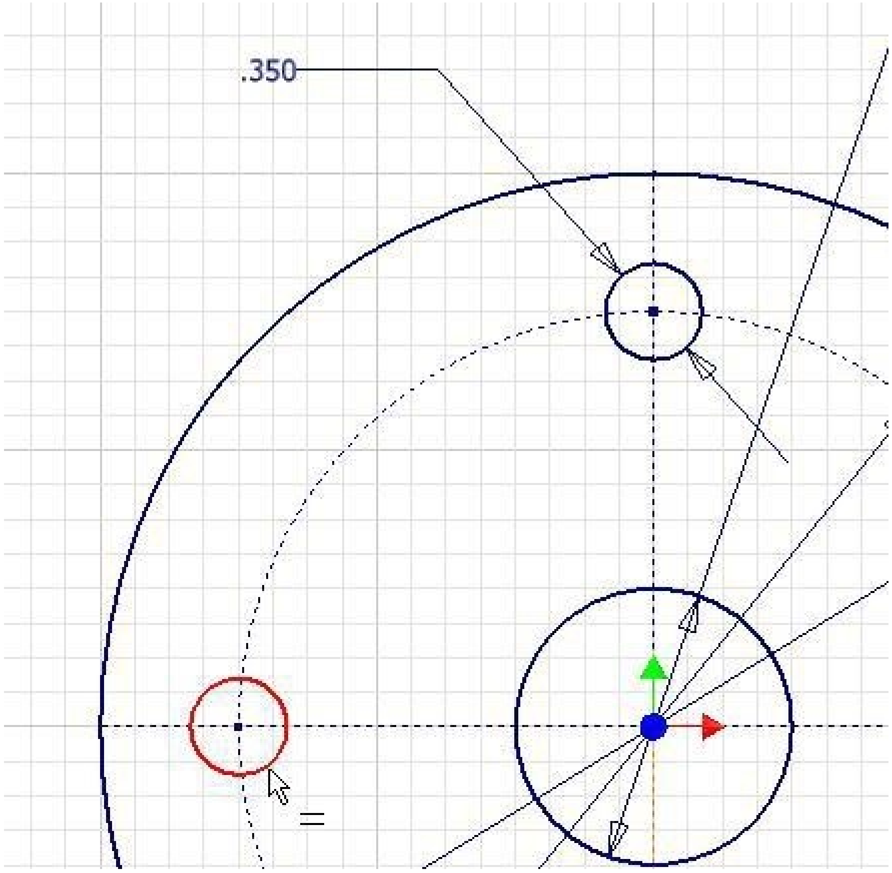


Figure Step 20B [Click to see image full size]

Step 21

Using what you learned in Step 20, constrain the additional 2 circles with the Equal constraint to equal the dimensioned circle. The constraint icons should appear similar to the figure. (Figure Step 21)

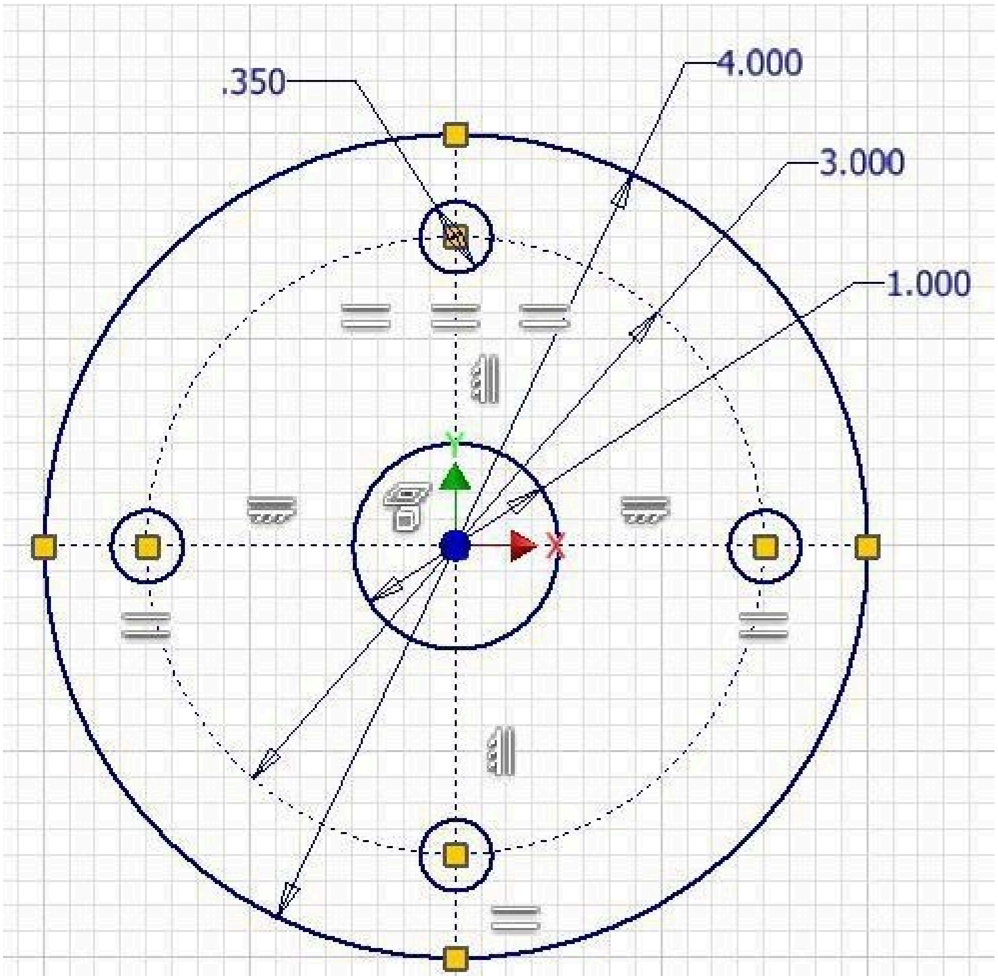


Figure Step 21 [Click to see image full size]

Step 22

Change the view to the Home view. The sketch should be fully constrained and display purple. (Figure Step 22)

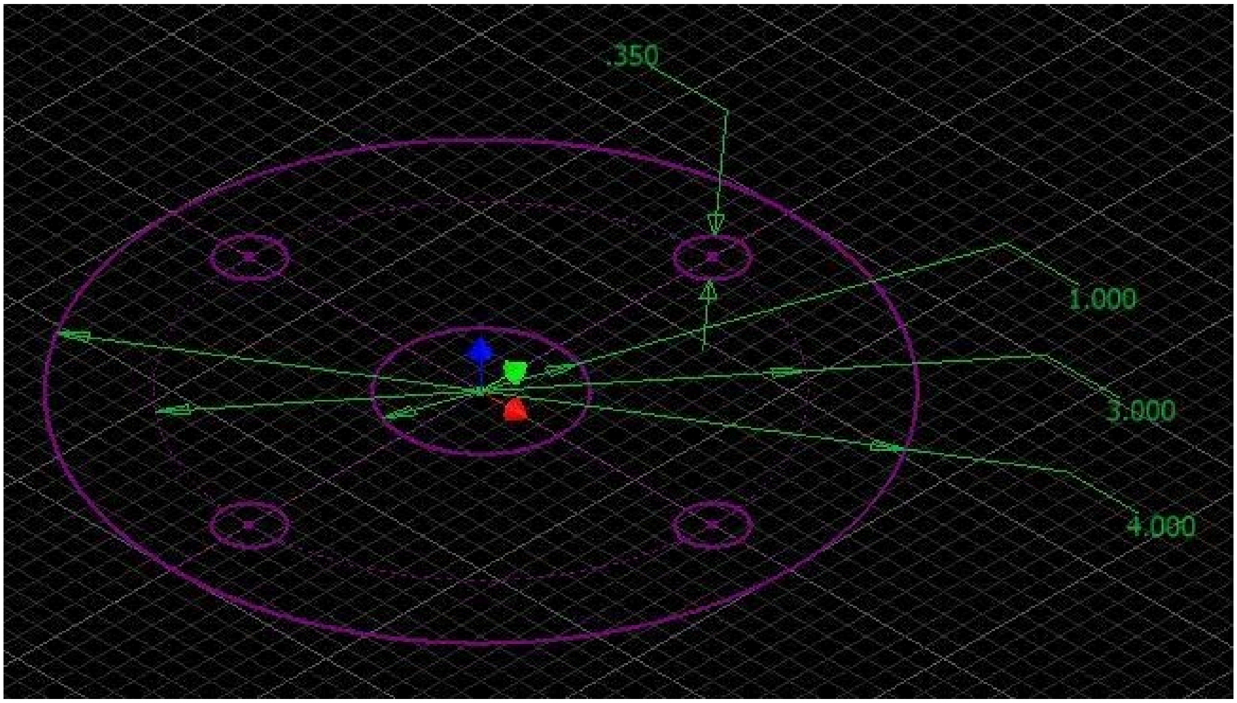
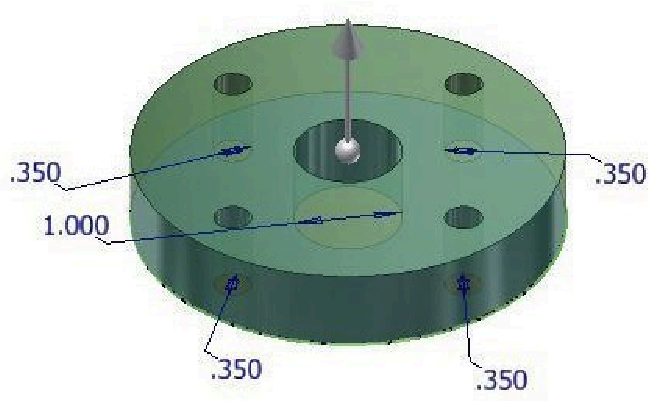


Figure Step 22 [Click to see image full size]

Step 23

Extrude the sketch as shown in the figure. (Figure Step 23A and 23B)



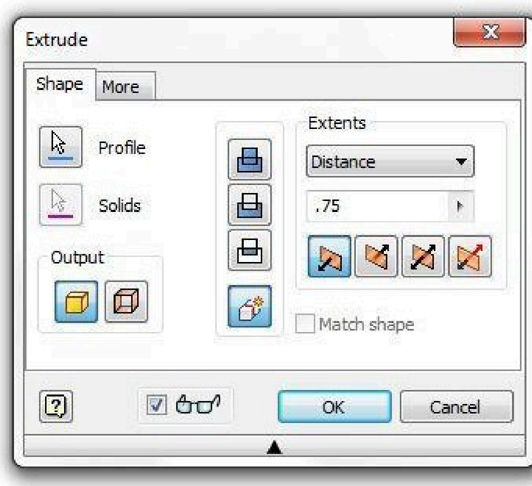


Figure Step 23A [Click to see image full size]

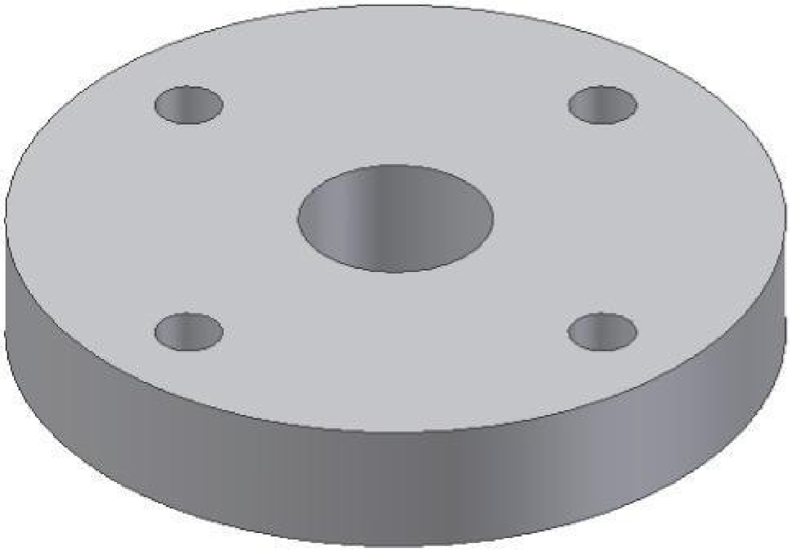


Figure Step 23B

Step 24

Start a new sketch on the top plane of the model. Using the OFFSET command, offset the outside diameter of the object to insert a 1.5 inch circle. Add the diameter dimension as shown in figure. (Figure Step 24)

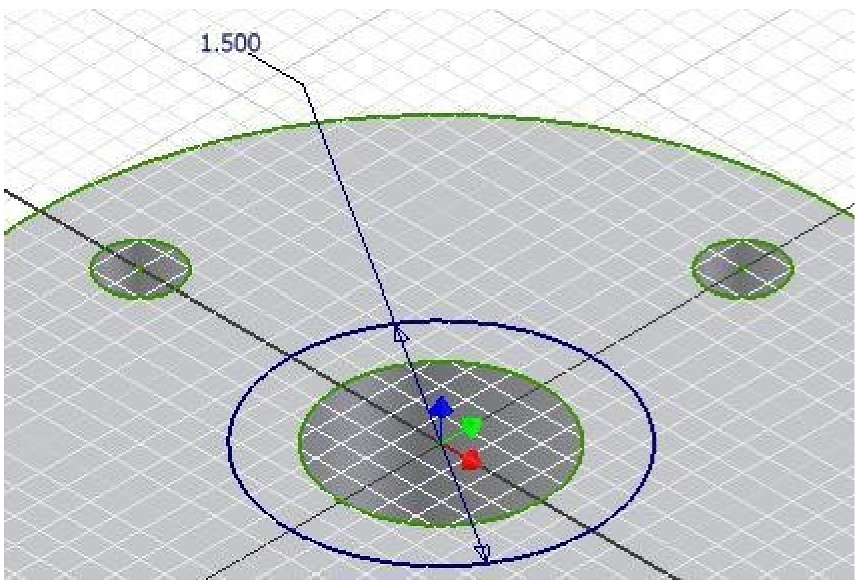


Figure Step 24

AUTHOR'S COMMENTS: Since you used the OFFSET command to draw the circle, the centre of the circle will be automatically constrained and the sketch will be fully constrained.

Step 25

Extrude the sketch as shown in figure. (Figure Step 25)

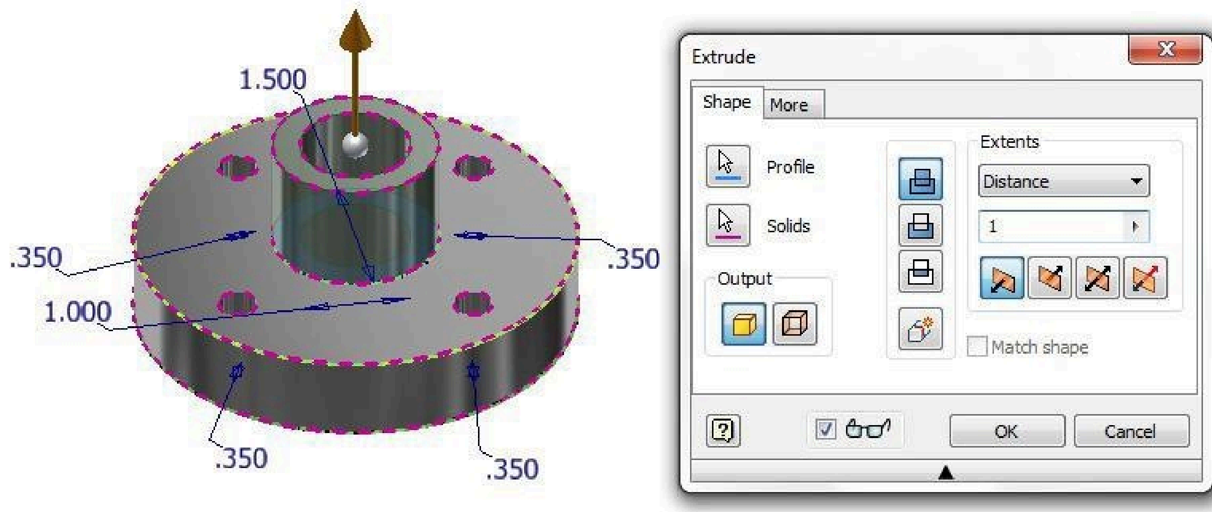


Figure Step 25 [Click to see image full size]

AUTHOR'S COMMENTS: Notice how the Join icon is enabled. This is because you are joining onto the existing solid.

Step 26

Change the colour to: Chrome -Polished Black. (Figure Step 26)

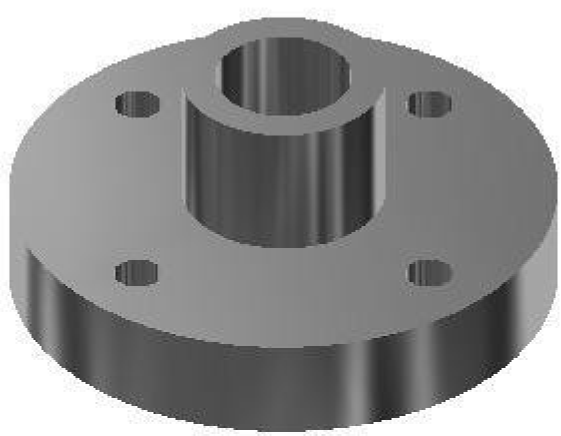


Figure Step 26

Step 27

Save and close the file.

Key Principles

Key Principles in Module 12

1. Construction objects are objects that are drawn in the sketch to assist you in completing the sketch but will be ignored by Inventor when the sketch is extruded or revolved. Geometrical and dimensional constraints can be applied to construction objects the same as they are to drawing Construction objects and drawing objects only differ in the properties of the objects. They are both drawn and manipulated the same way in a 2D sketch.
2. To select more then one object at time, hold down either the CTRL or SHIFT key while selecting the objects.

Lab Exercise 12-1

Time allowed: 45 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab Lab 12-1	Inventor Course	Millimeters	Metric-Modules Part (mm).ipt	Aluminum – Flat	N/A

Step 1

Project the Center Point onto the Base plane.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude them to produced the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. (Figure Step 2A and 2B)

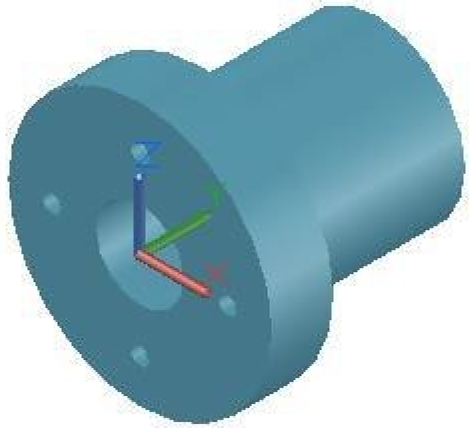


Figure Step 2A
Solid Model – Home View

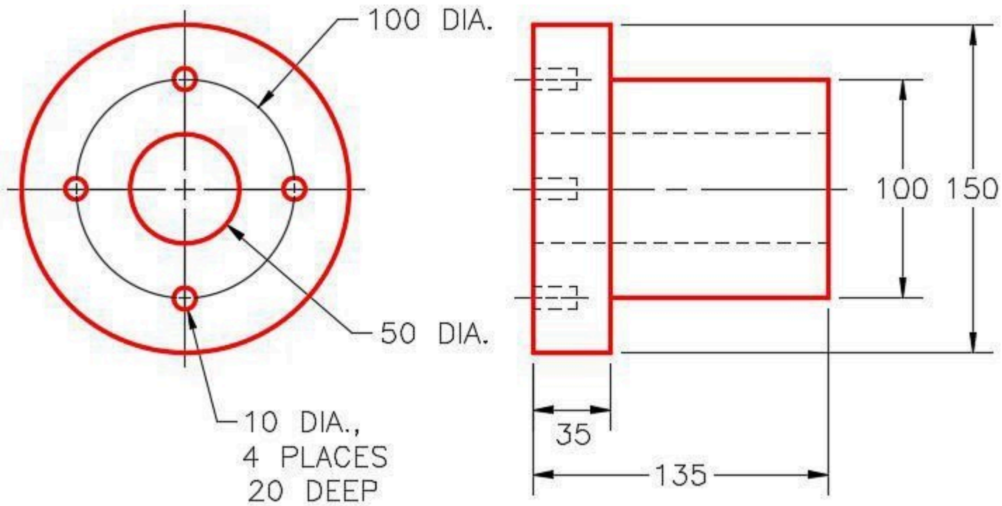


Figure Step 2B
Dimensioned Multiview Drawing [Click to see image full size]

Step 3

Apply the colour shown above. (Figure Step 3A and 3B)

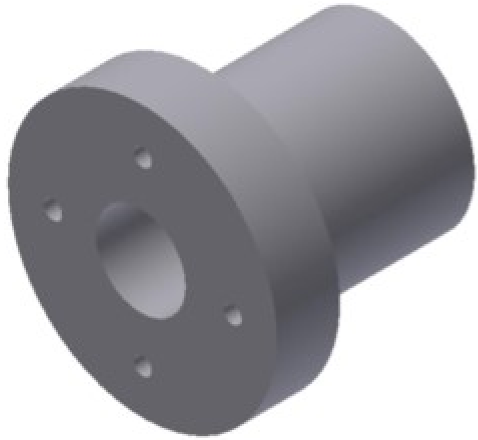


Figure Step 3A
Solid Model – Home View

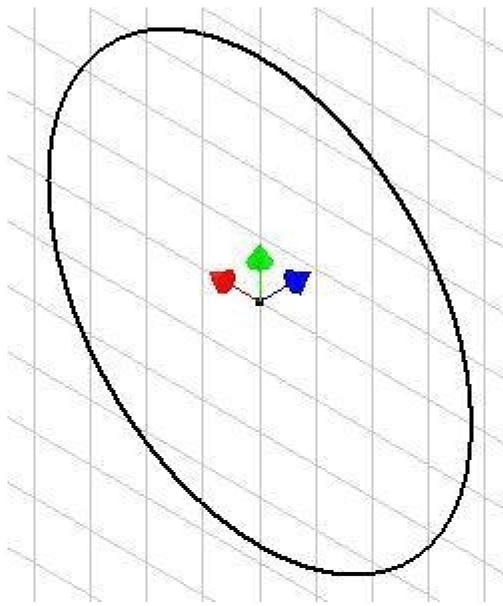
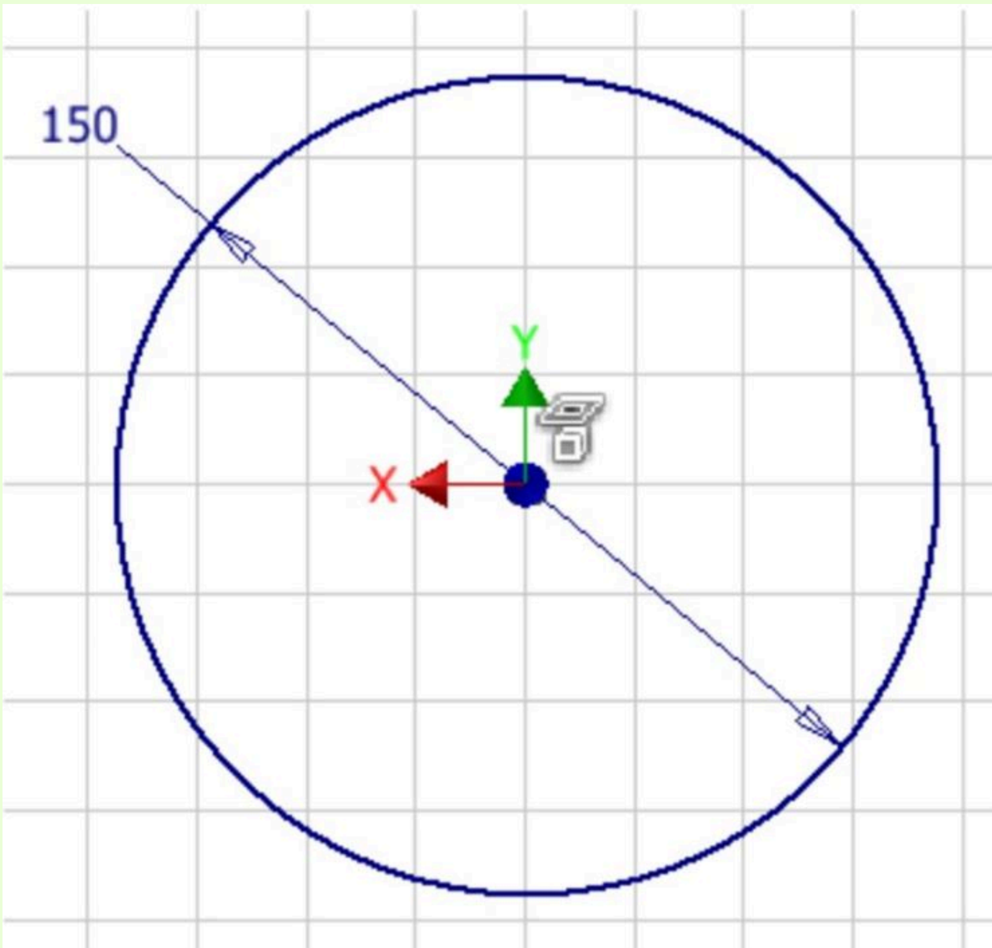


Figure Step 3B
Suggested Base Sketch –
Front – XZ Plane

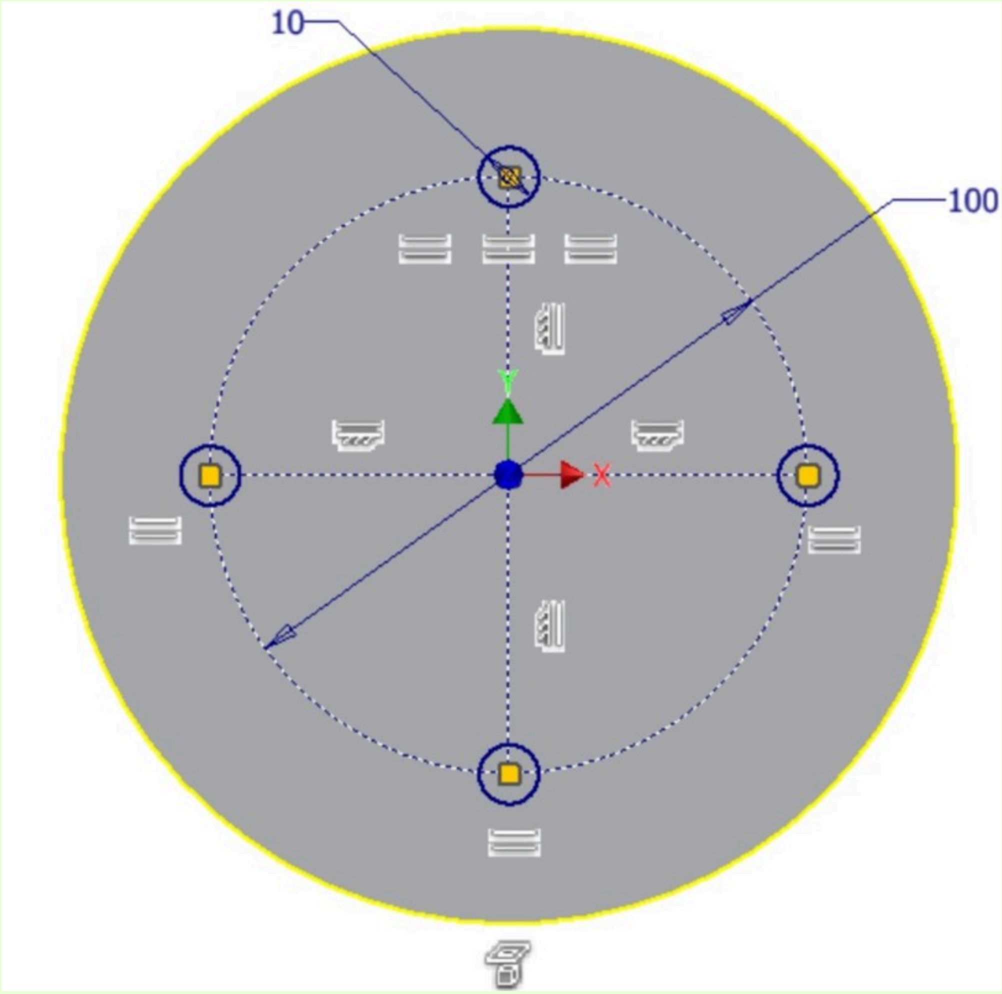
Hint:

The small holes do NOT go all the way through the flange.

AUTHOR'S GEOMETRIC CONSTRAINTS: The following figures shows the sketch's construction method plus geometric and dimensional constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketch using a different construction method and constraints, that is what is important. You may want to compare your construction method and constraints with the authors.



Base Sketch [Click to see image full size]



Sketch 2 [Click to see image full size]

Lab Exercise 12-2

Time allowed: 45 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab Lab 12-2	Inventor Course	Inches	English-Modules Part (in).ipt	White	N/A

Step 1

Project the Center Point onto the base plane.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude them to produced the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. (Figure Step 2A and 2B)

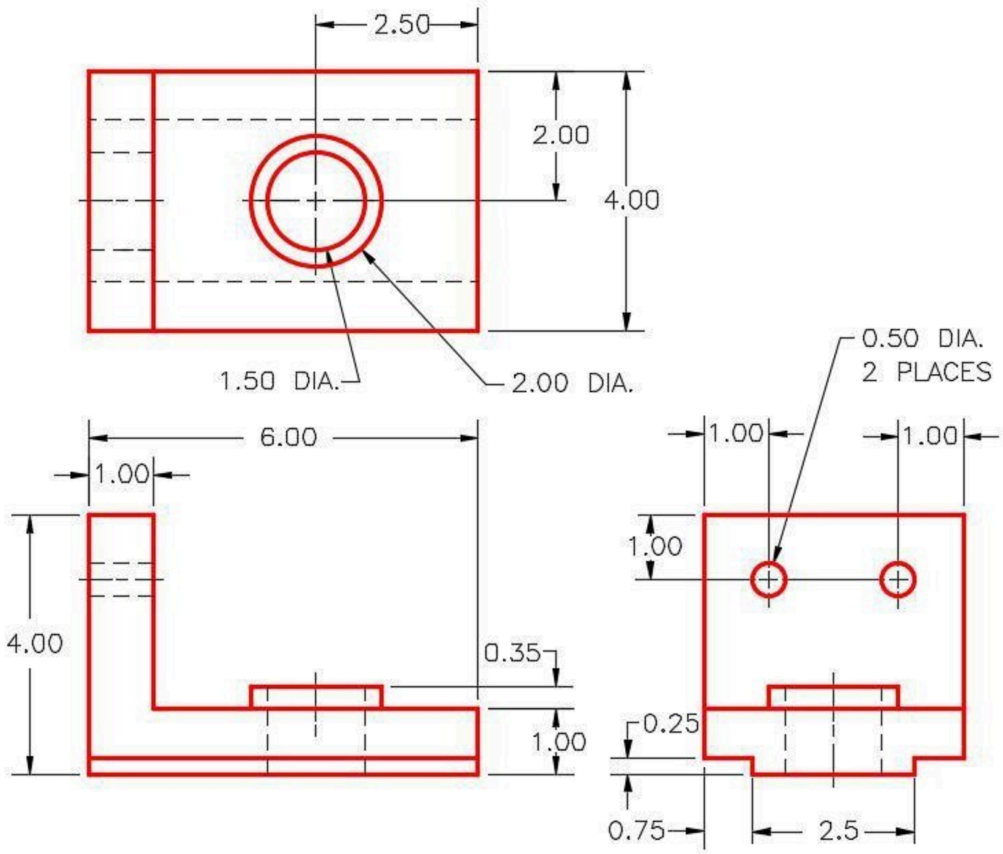


Figure Step 2A
Dimensioned Multiview Drawing [Click to see image full size]

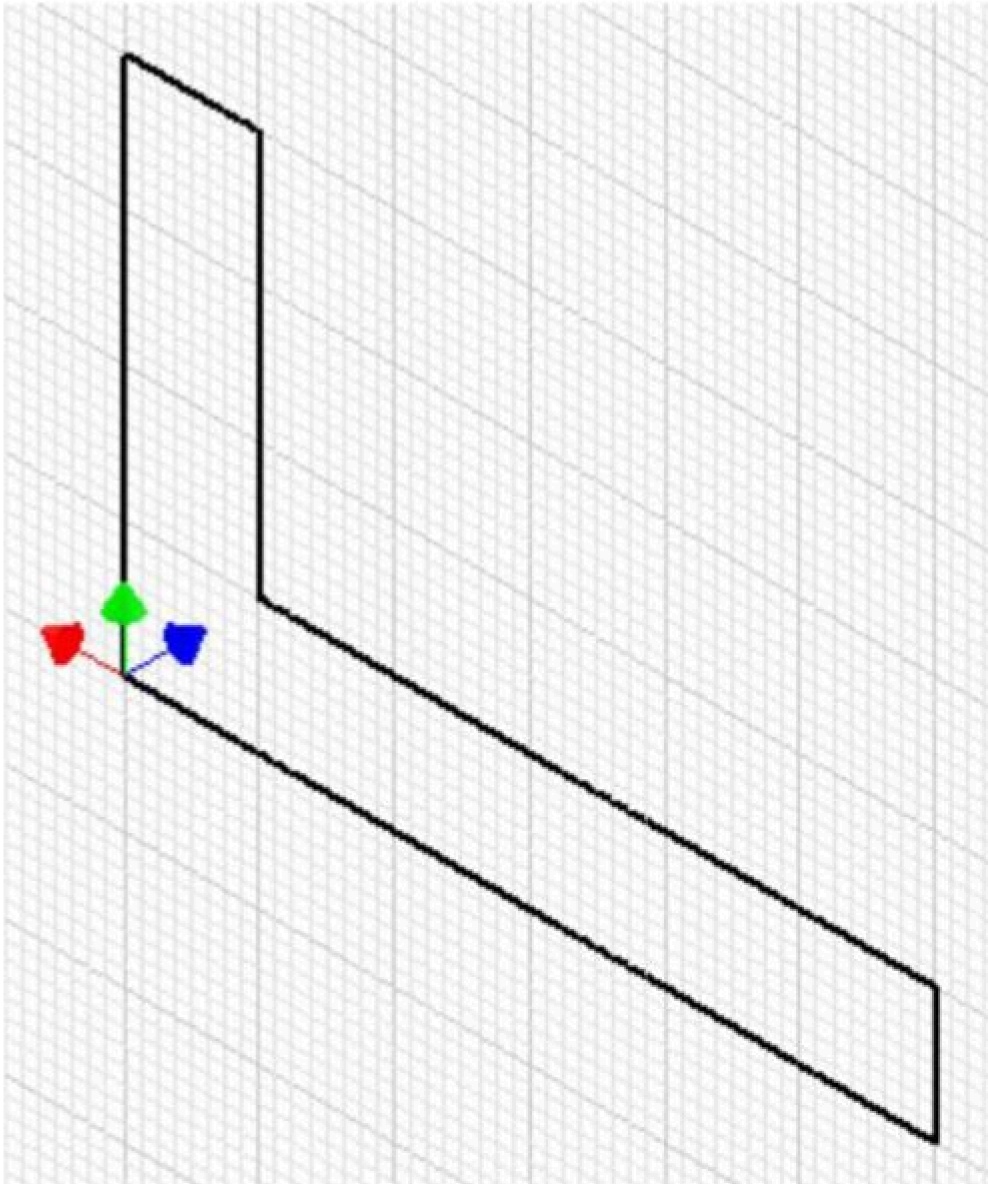


Figure Step 2B
Suggested Base Sketch –
Front – XZ Plane [Click to see image full size]

Hint

Draw the base sketch on the Front view or XZ plane.

Step 3

Apply the colour shown above. (Figure Step 3A and 3B)

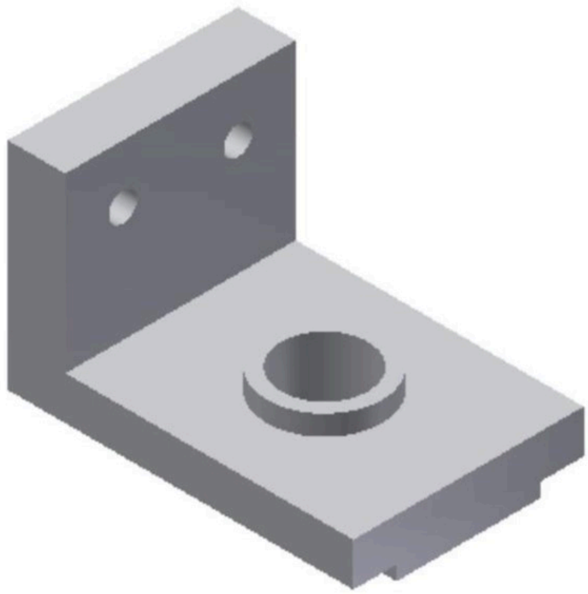


Figure Step 3A
Completed Solid Model – Home View [Click to see image full size]

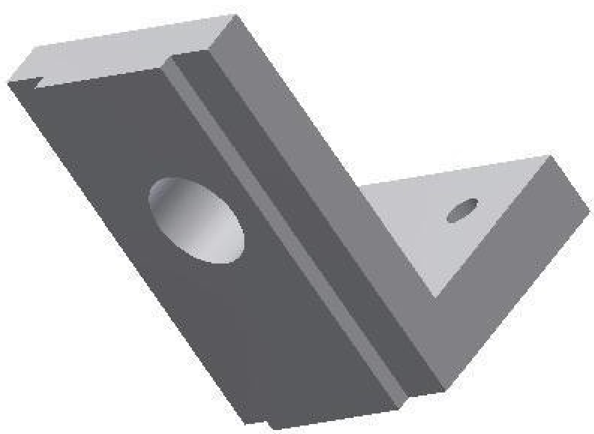


Figure Step 3B
Completed Solid Model – Bottom View [Click to see image full size]

Module 13 Arcs

Learning Outcomes

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

1. Describe the geometry of an arc and how arcs are drawn in Inventor.
2. Describe how to snap to midpoints of lines.
3. Apply the CENTER POINT ARC command to draw arcs in 2D sketches.

Drawing Arcs

There are three commands available to draw arcs in Inventor. In this module, drawing arcs using the CENTER POINT ARC command will be taught. Similar to the CIRCLE command, the CENTER POINT ARC command also requires you to select the centre point and the radius. Drawing arcs additionally require you to locate the start point and end point or the arc. Arcs can be constructed clockwise or counterclockwise.

Geometry Lesson: Arcs

An *ARC* is defined as an open curve in which all points are the same distance from its centre point. Study the drawings in Figure 13-1 and 13-2.for a description of the geometry of an arc.

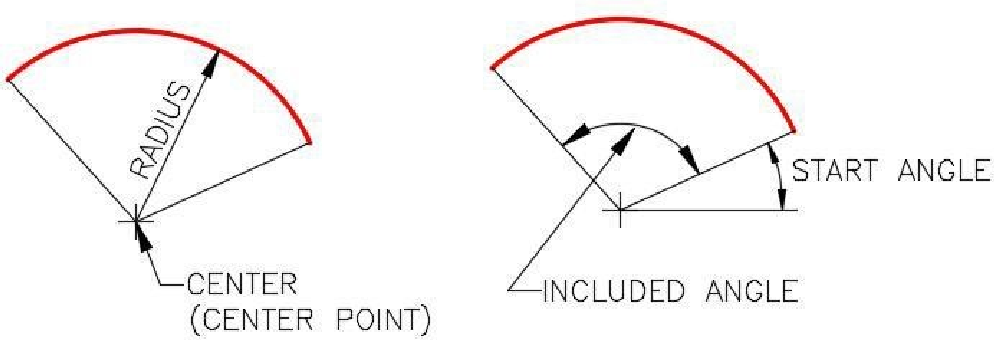


Figure 13-1
Geometry of an Arc – Part 1

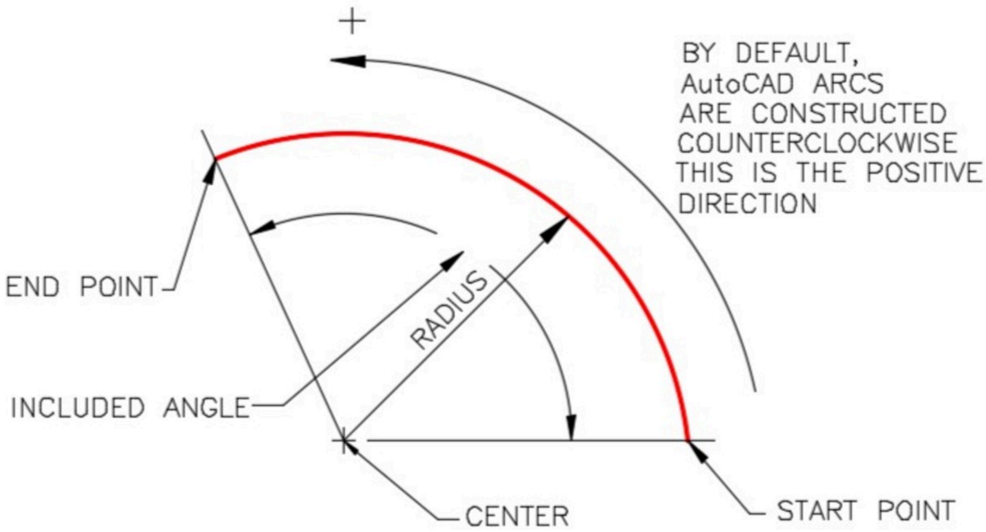
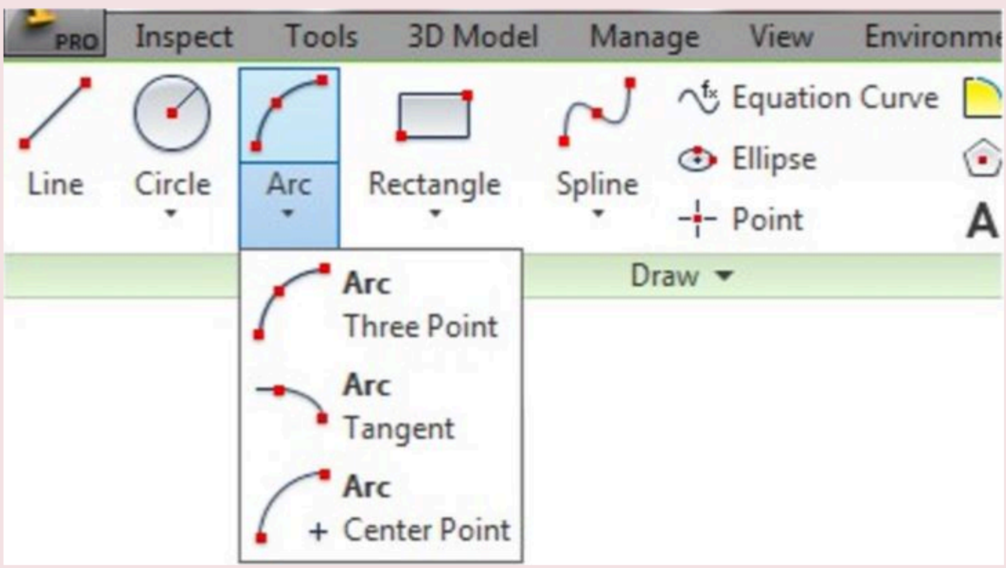


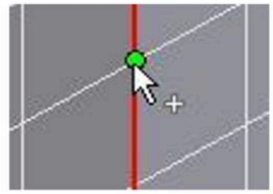
Figure 13-2
Geometry of an Arc – Part 2

Inventor Command: CENTER POINT ARC

The CENTER POINT ARC command is used to draw an arc by entering the location of its centre point, radius, start point and end point.

Shortcut: **A**



Snap Symbols		
Mode	**Icon**	**How it appears in Inventor**
Midpoint (Snaps to the midpoint of an object)		

Geometrical Constraint - Coincident		
Constraint	**Icon**	**Definition**
Coincident		A coincidence constraint indicates that two points or endpoints of an object are exactly at the same XY location.

WORK ALONG: Drawing Models with Arcs

Step1

Check the default project and if necessary, set it to Inventor Course.

Step 2

Enter the NEW command to start a new part file using the template: Metric-Modules Part (mm).ipt.

Step 3

Save the file with the name: Inventor Workalong 13-1. (Figure Step 3A and 3B)

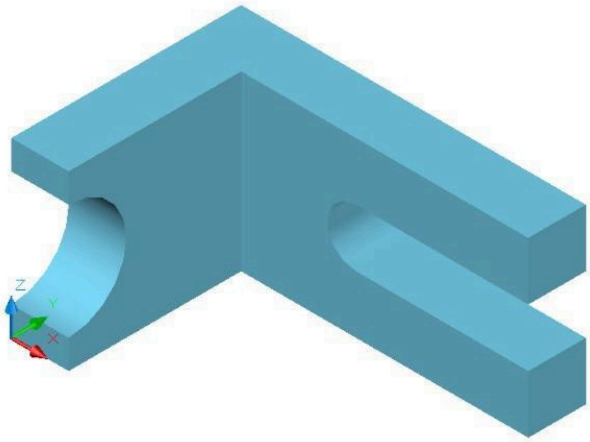


Figure Step 3A
3D Model – Home View [Click to see image full size]

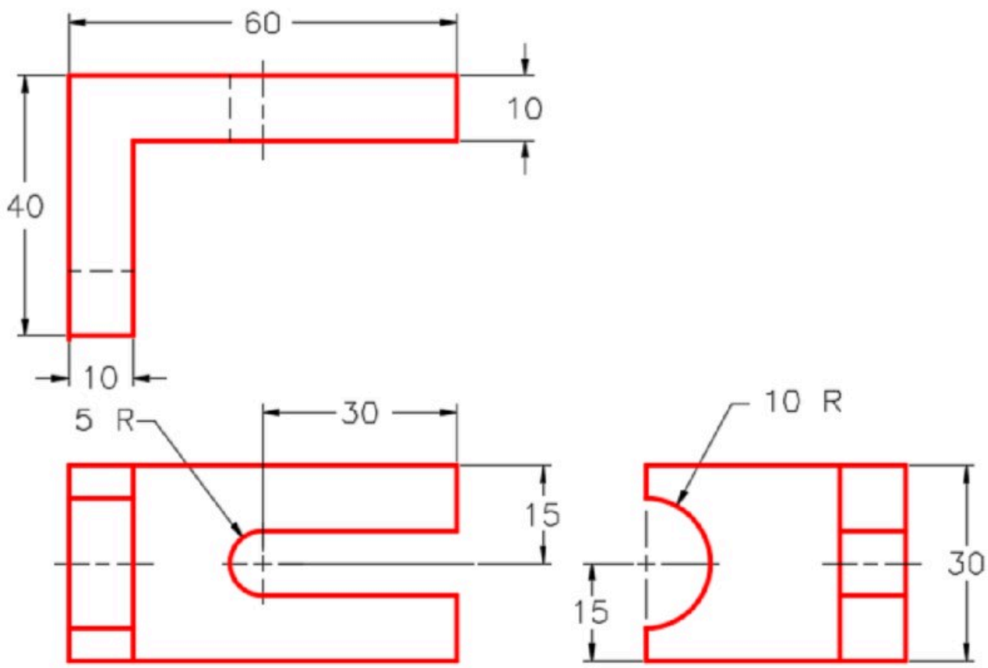


Figure Step 3B
Dimensioned Multiview Drawing [Click to see image full size]

USER TIP: When drawing arcs, do your best to tie the centre point, the start point, and the end point to existing geometry. It will decrease the number of dimensions that are required to constrain the arc.

Step 4

Edit Sketch1. Project the Center Point onto the sketch.

Step 5

Draw a sketch of the Top view of the model. Apply all of the geometrical and dimensional constraints to fully constrain it. (Figure Step 5)

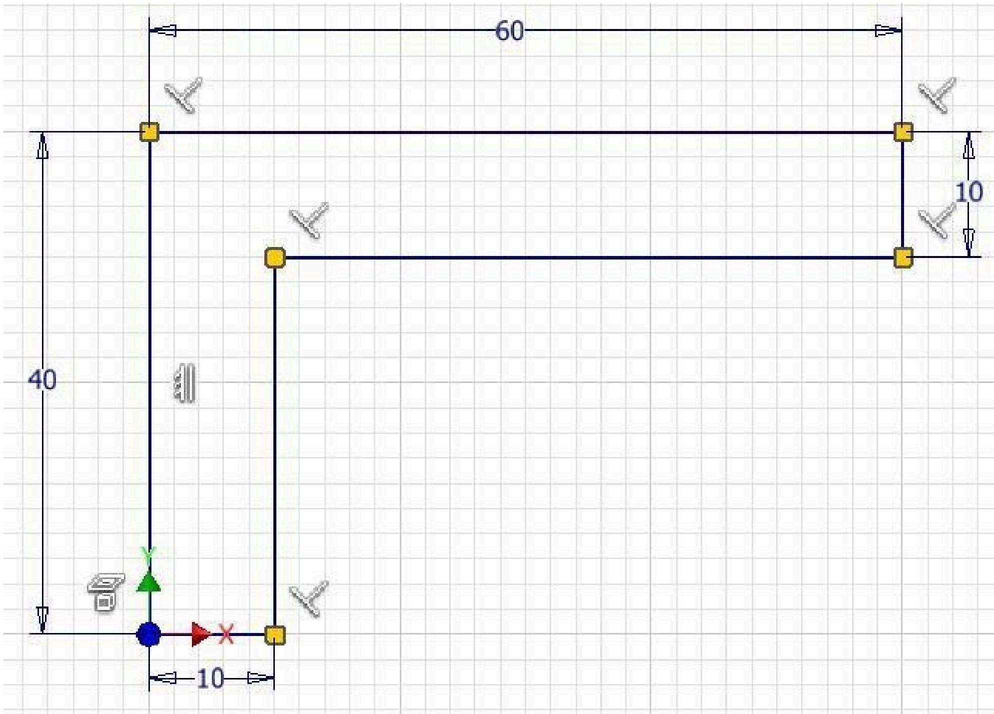


Figure Step 5 [Click to see image full size]

AUTHOR'S COMMENTS: Your geometrical and dimensional constrains may not match the figure exactly. Just ensure that your sketch is fully constrained.

Step 6

Extrude the sketch in the positive Z direction. (Figure Step 6)

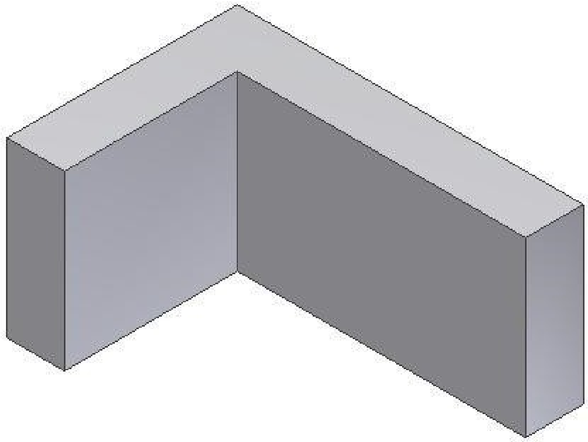


Figure Step 6

Step 7

Start a new sketch on the right side as shown in the figure. (Figure Step 7)

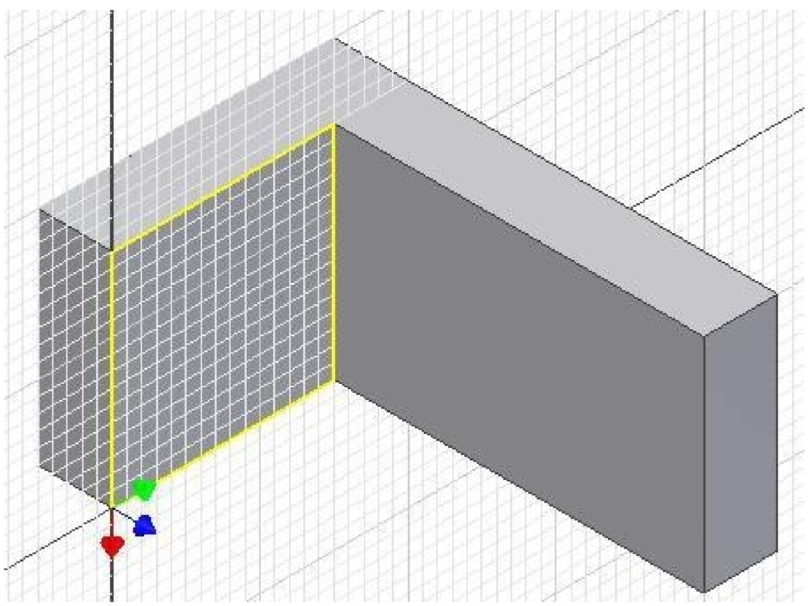


Figure Step 7

Step 8

Enter the CENTER POINT ARC command and when prompted for the centre point, right click the mouse. In the Right-click menu, select Midpoint. (Figure Step 8)

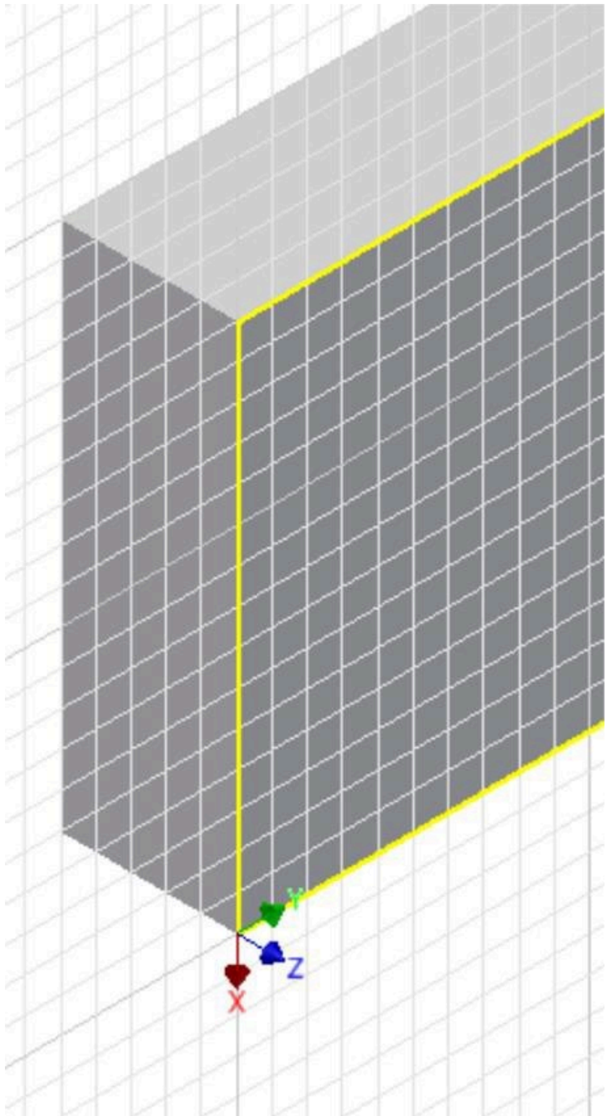


Figure Step 8

Step 9

Move the cursor onto the vertical edge. When the green snap icon appears, click the left mouse button. (Figure Step 9)

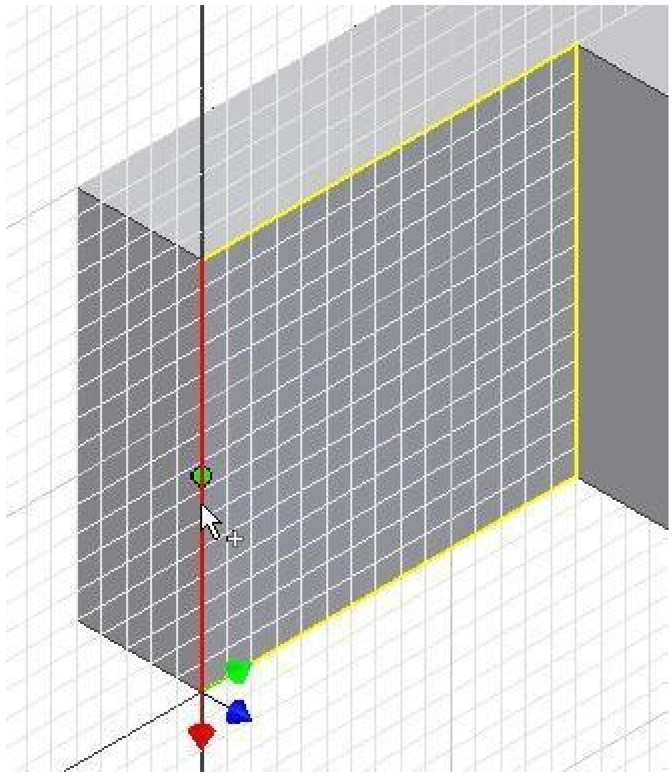


Figure Step 9

Step 10

Move the cursor approximately 10 mm along the edge and when the Snap onto icon displays, click the mouse. (Figure Step 10)

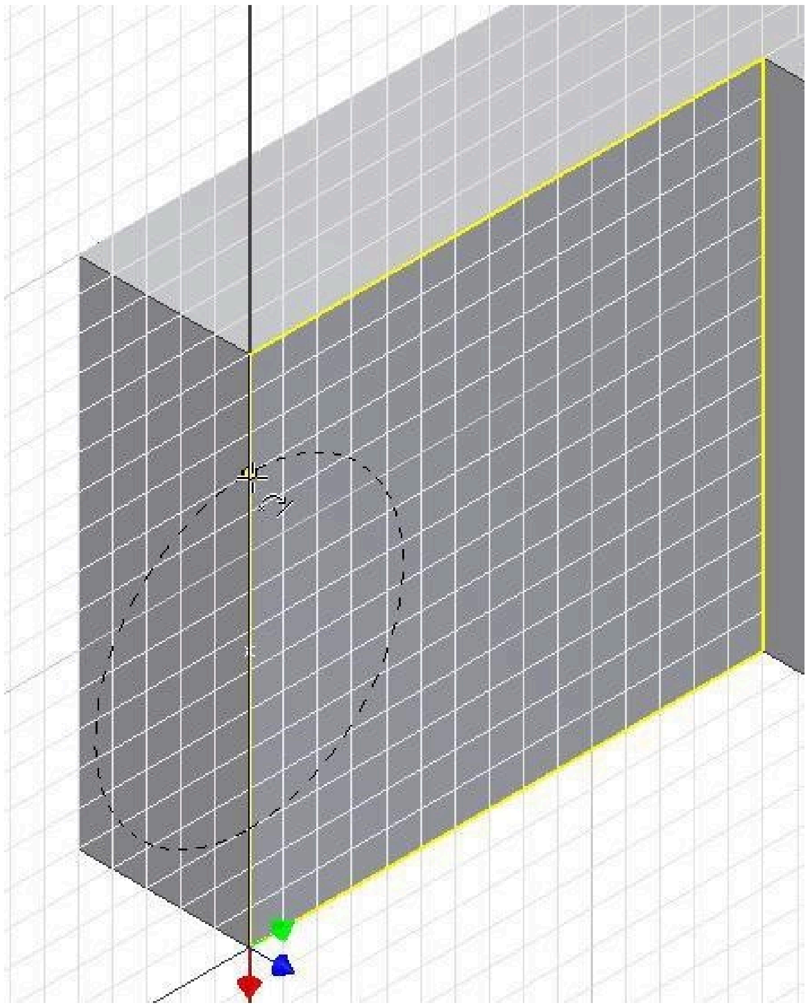


Figure Step 10

Step 11

Move the cursor to the other side of the centre and when the Snap onto icon displays, click the mouse. (Figure Step 11)

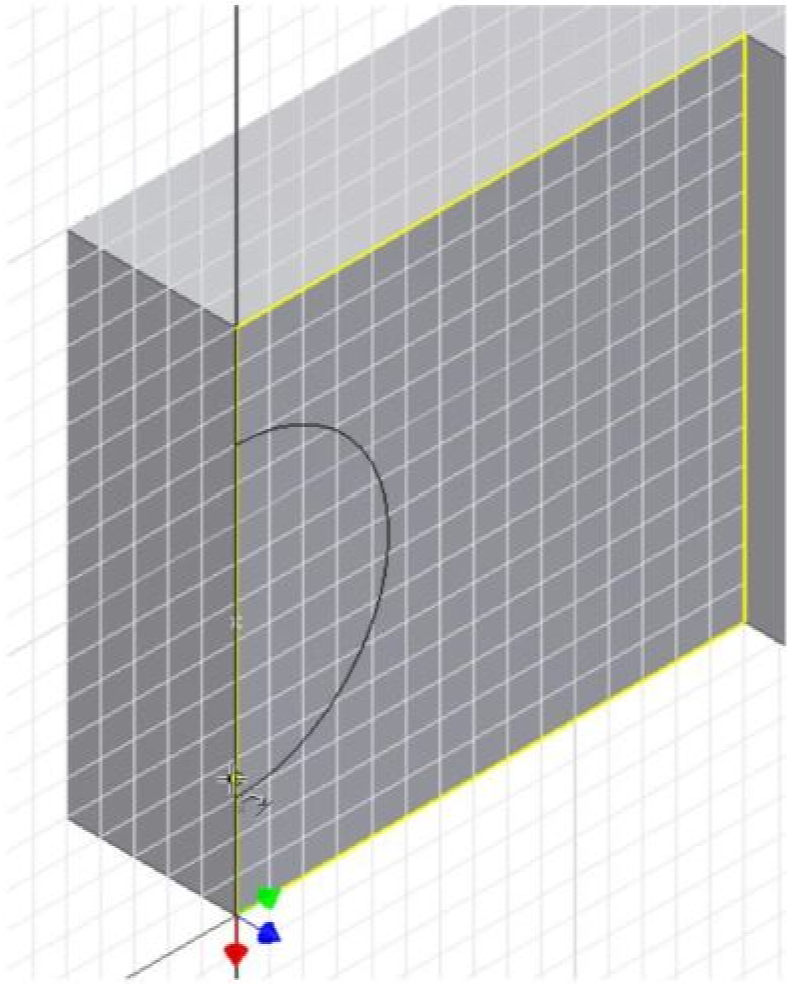


Figure Step 11

Step 12

Press F8 to display the constraint icons. (Figure Step 12)

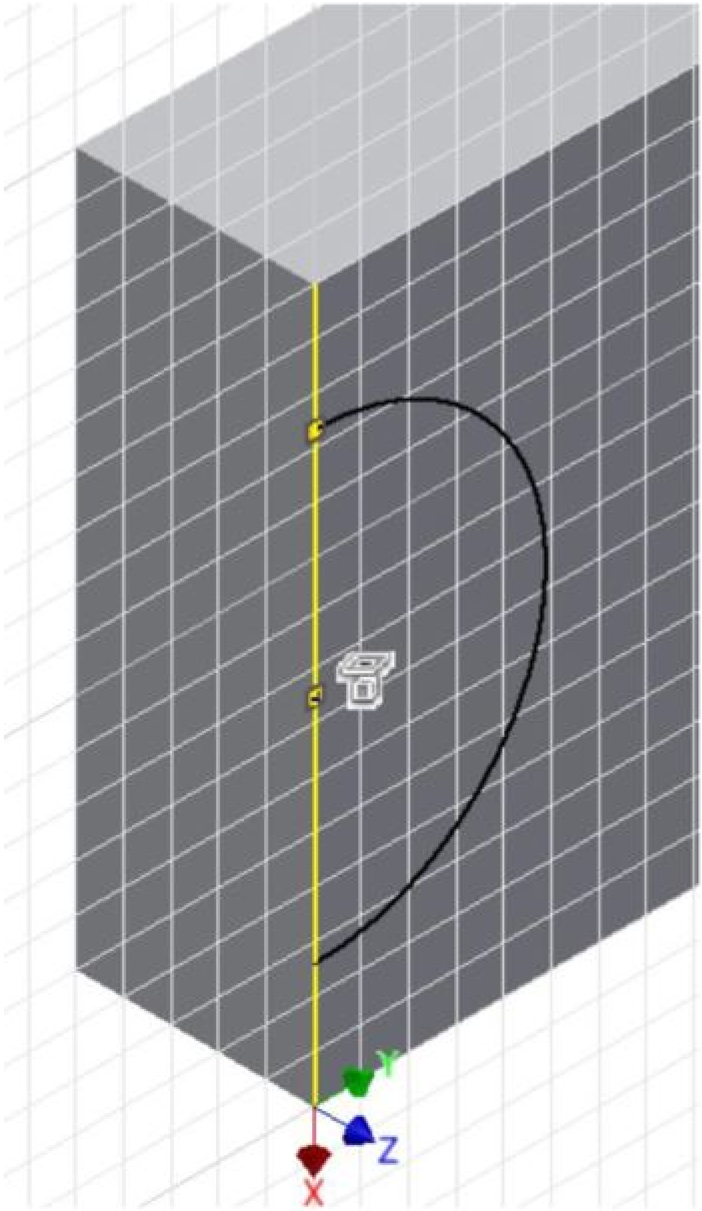


Figure Step 12

Step 13

Move the cursor onto the Coincident icon and note the display of two coincident icons. (Figure Step 13)

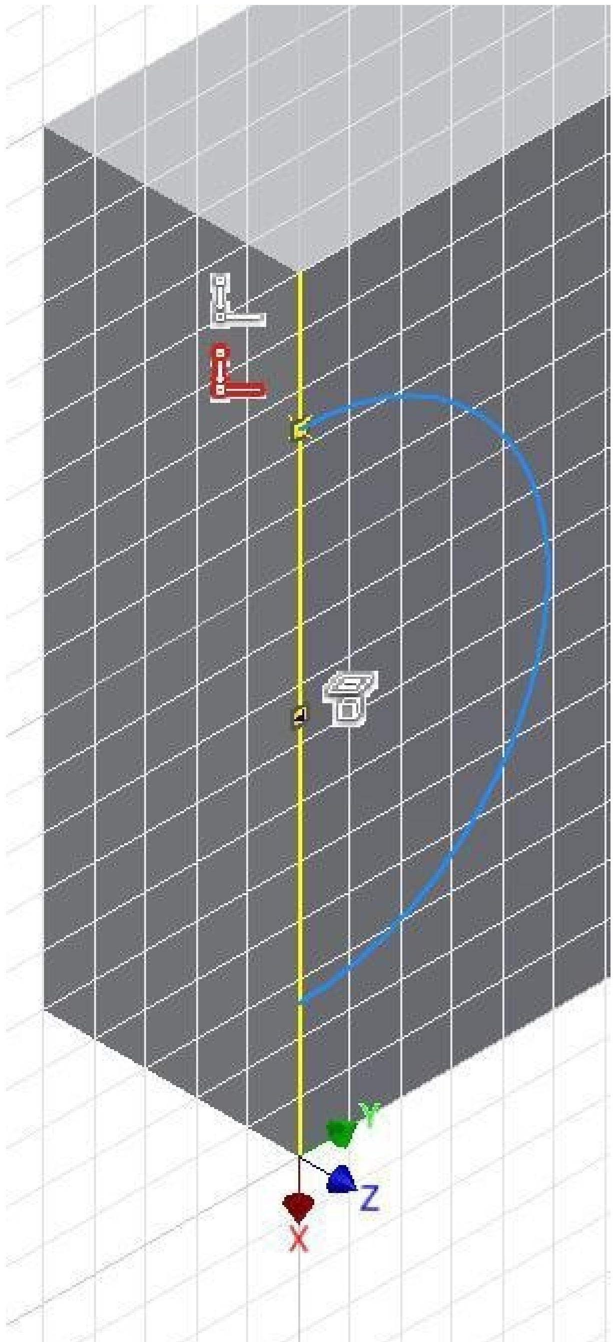


Figure Step 13

AUTHOR'S COMMENTS: When two constraint icons display, it means that the endpoint of the arc is exactly on the line and the line is exactly on the endpoint of the arc.

AUTHOR'S COMMENTS: Note that the bottom endpoint of the arc is not constrained with a Coincident constraint.

Step 14

In the Graphic window, right click the mouse. In the Right-click menu, click Create Constraint and then Coincident. (Figure Step 14)

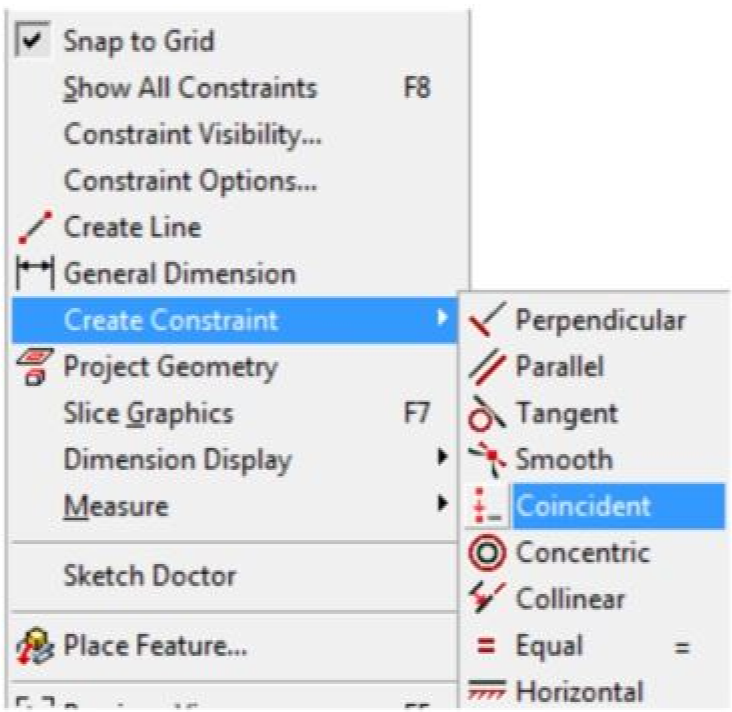


Figure Step 14

Step 15

Select the edge for the first point. (Figure Step 15)

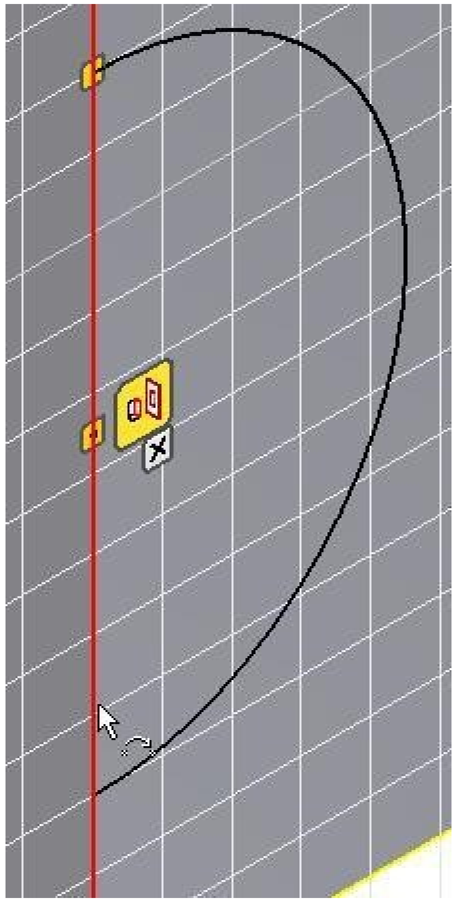


Figure Step 15

Step 16

For the second point, move the cursor onto the endpoint of the arc. When a small red point will displays, click the mouse. (Figure Step 16)

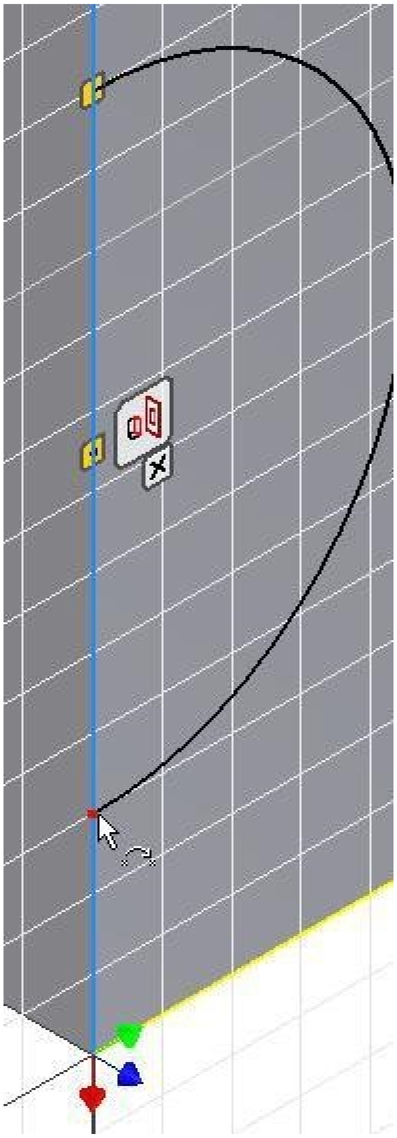


Figure Step 16

Step 17

Press F8. Locate the cursor onto the Coincident icon and note the display of two coincident icons. (Figure Step 17)

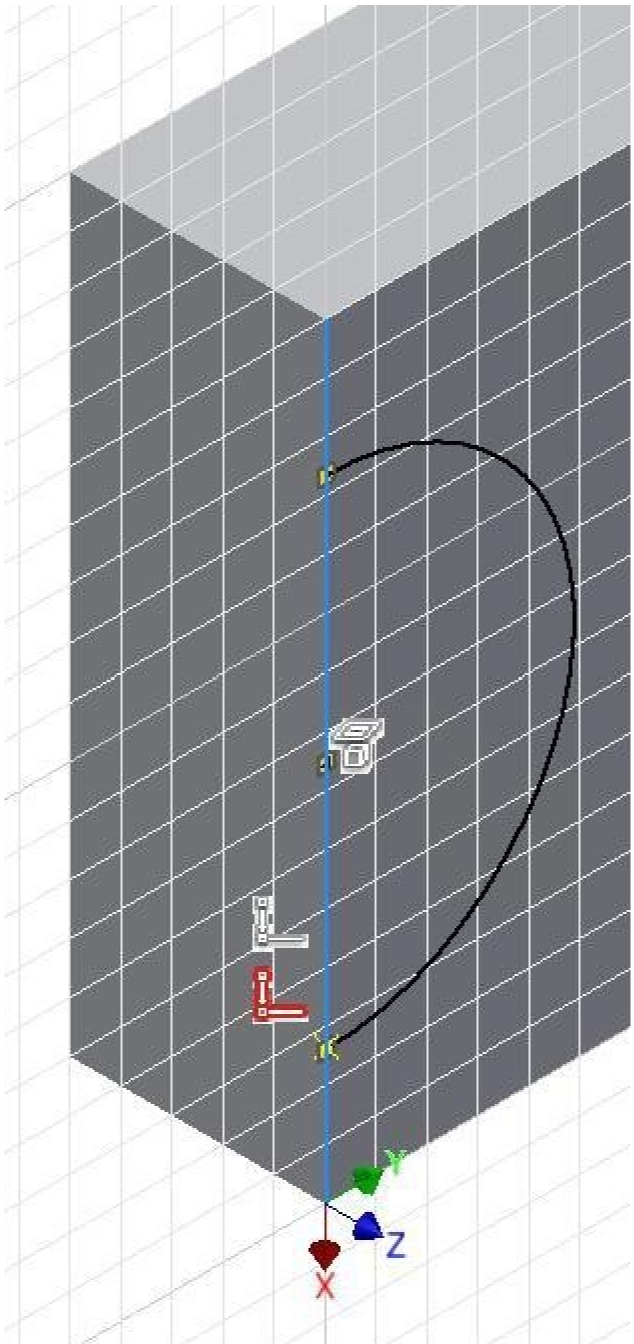


Figure Step 17

AUTHOR'S COMMENTS: Both ends of the arc are now geometrically constrained to the edge of model.

Step 18

The arc should now display purple indicating it is fully constrained. (Figure Step 18)

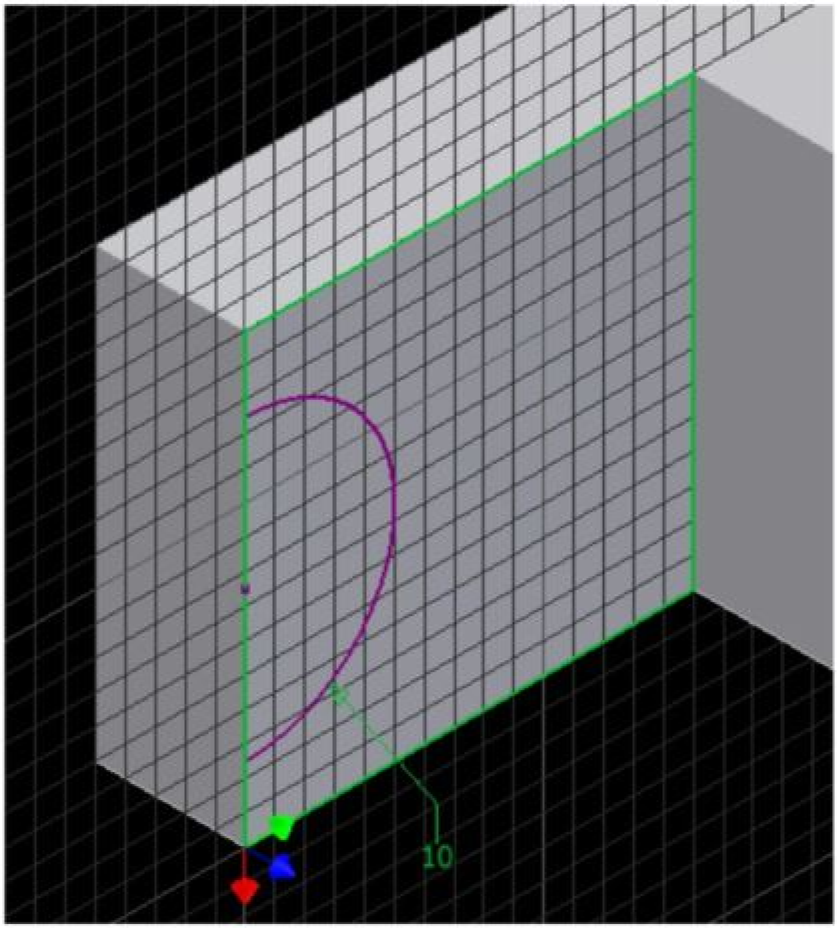


Figure Step 18

Step 19

Finish the sketch and extrude the arc by using the Cut option. (Figure Step 19)

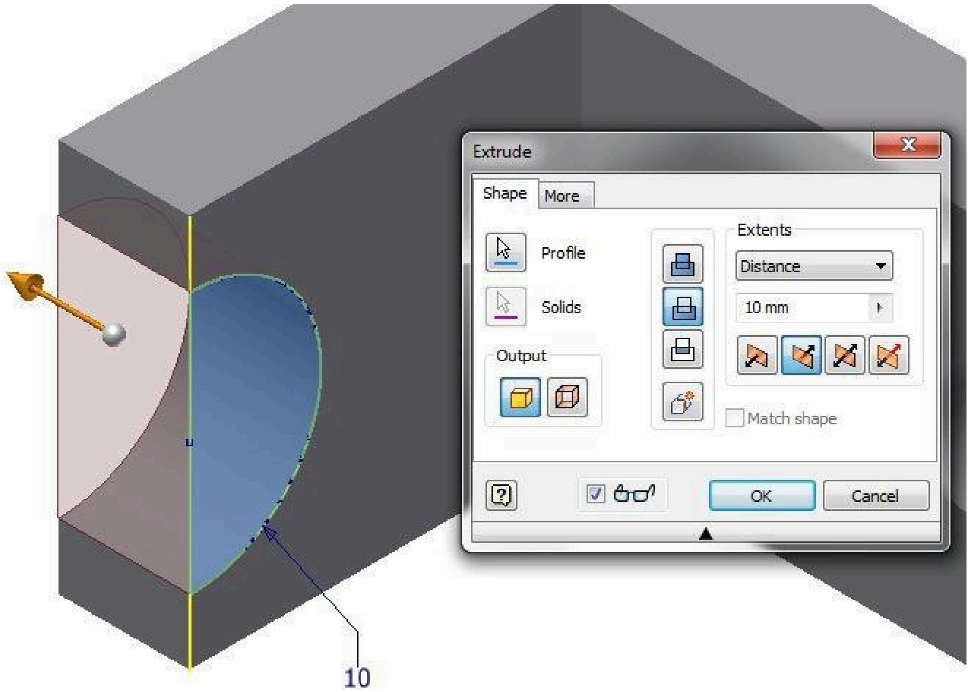


Figure Step 19 [Click to see image full size]

Step 20

Start a new sketch on the front side. (Figure Step 20)

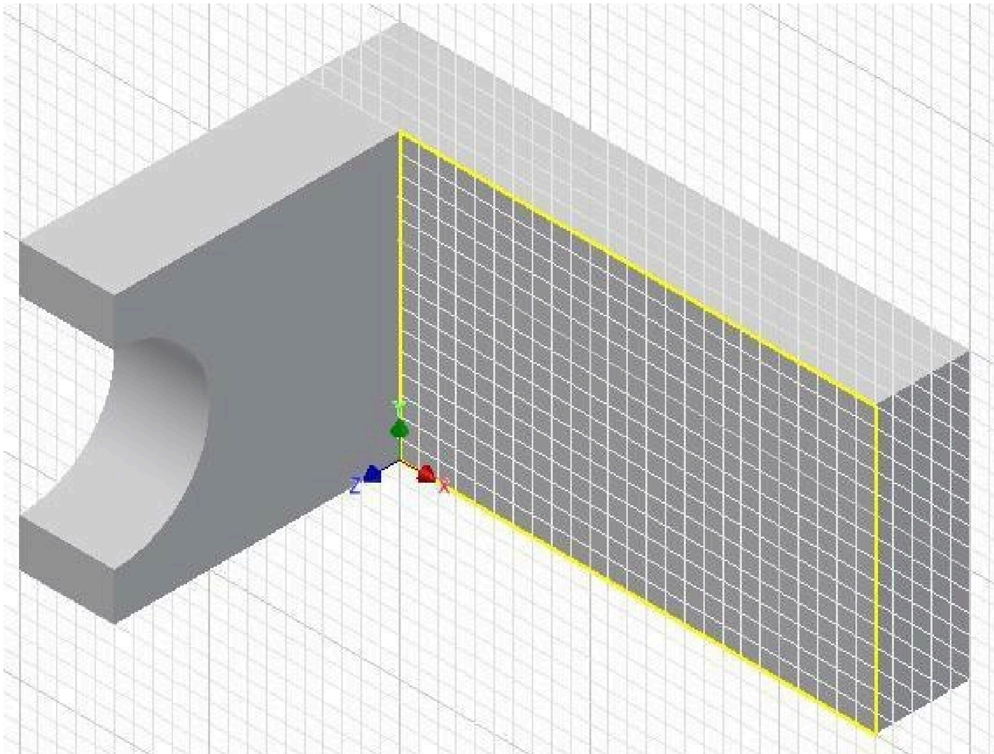


Figure Step [Click to see image full size]

Step 21

Draw a line perpendicular from the midpoint of the edge. (Figure Step 21)

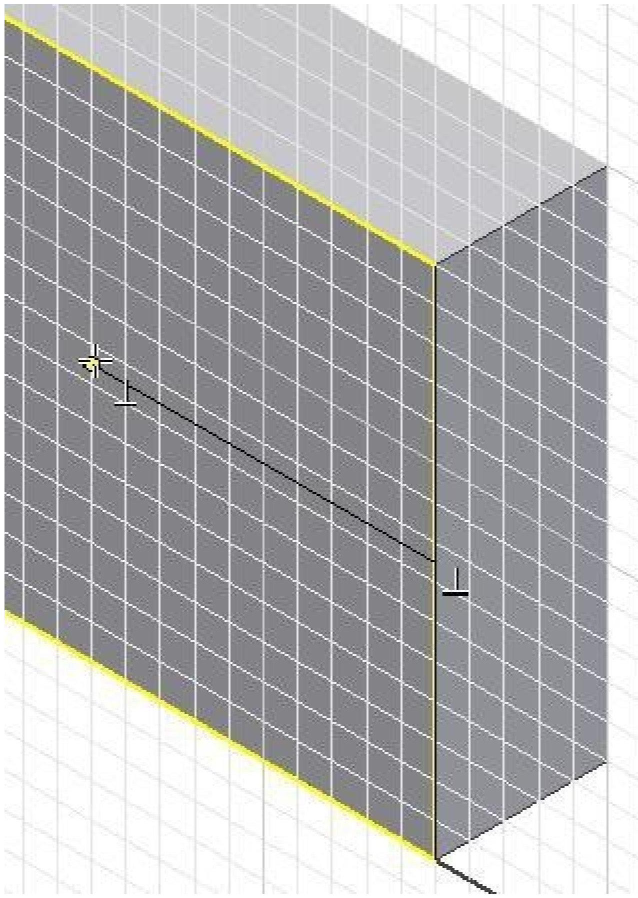


Figure Step 21

AUTHOR'S COMMENTS: Ensure that you snap to the midpoint of the edge.

Step 22

Change the line to a construction line by selecting the line and then click the Construction icon. (Figure Step 22A, 22B, and 22C)

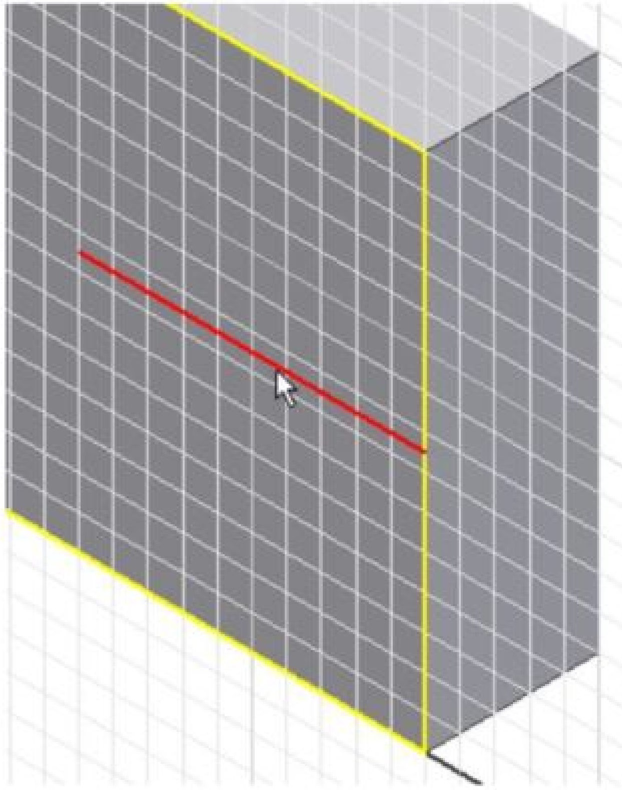


Figure Step 22A

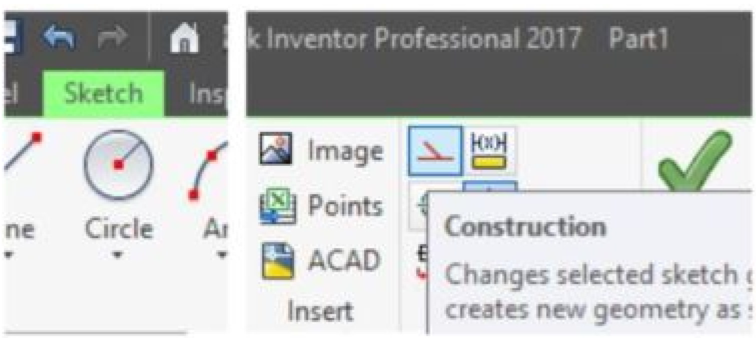


Figure Step 22B

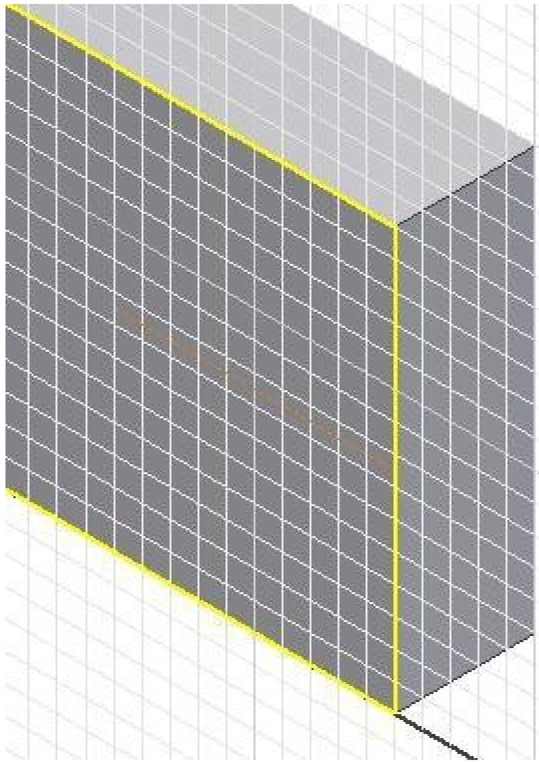


Figure Step 22C

Step 23

Offset the construction line on each side. (Figure Step 23)

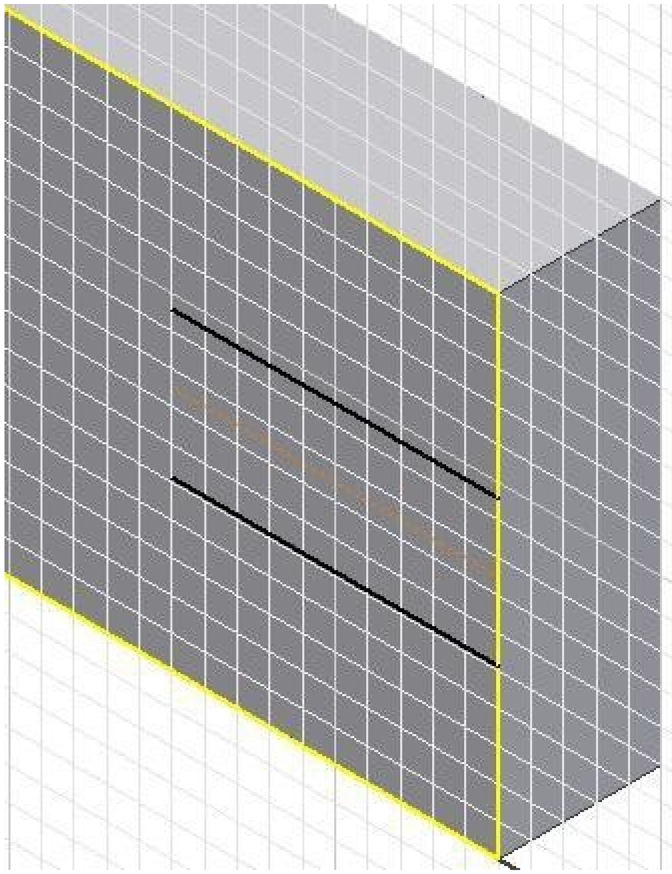


Figure Step 23

Step 24

Insert two dimensions. (Figure Step 24)

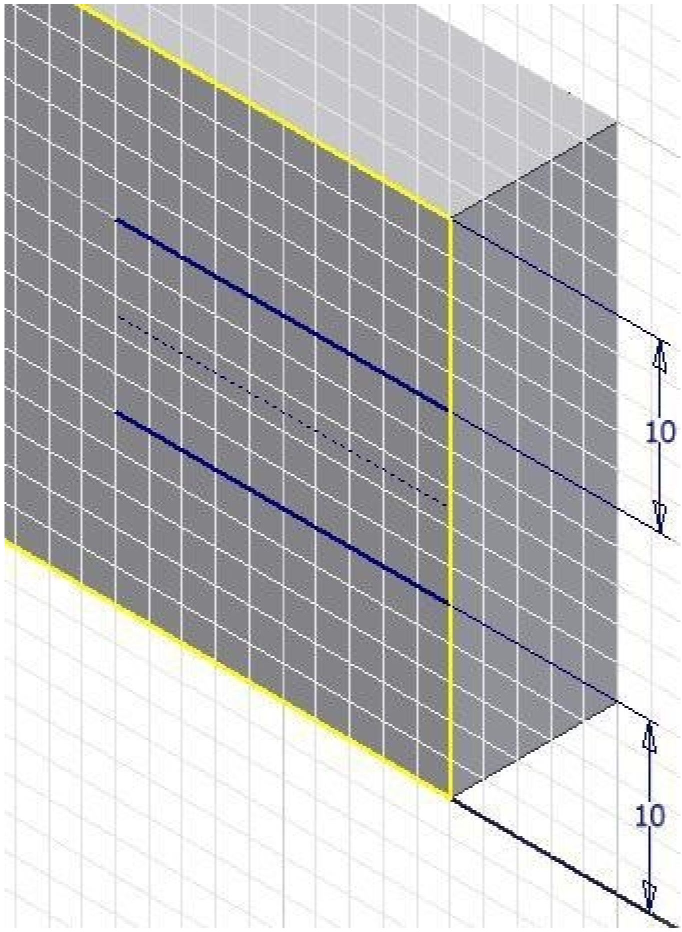


Figure Step 24

Step 25

Draw an arc locating the centre at the end of the construction line. (Figure Step 25)

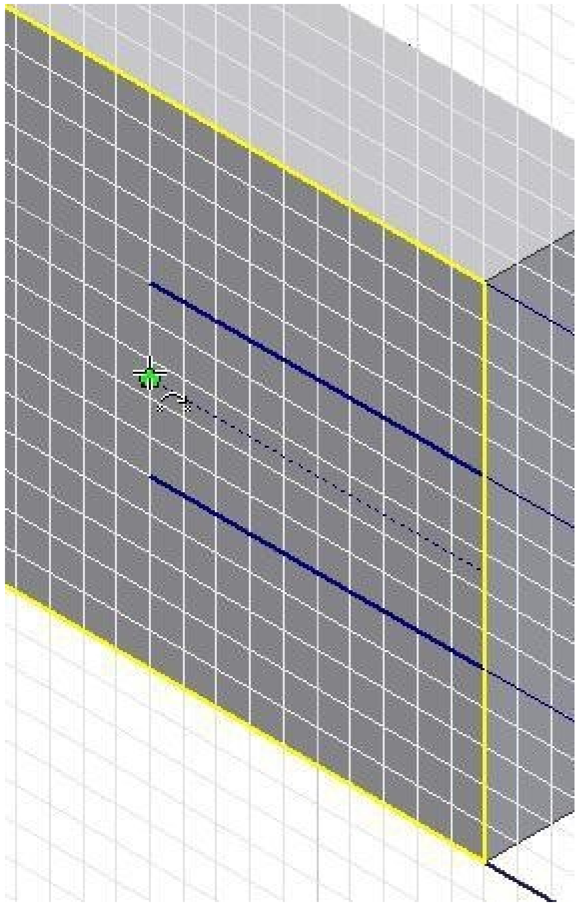


Figure Step 25

Step 26

Insert the required dimensions to constrain the sketch. (Figure Step 26)

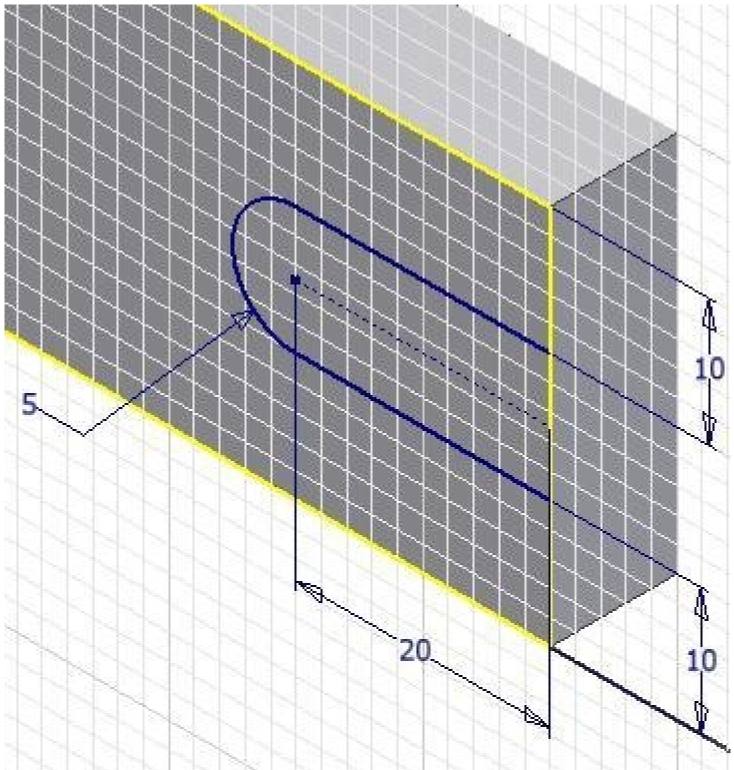


Figure Step 26

Step 27

Enable the display of the constraint icons. (Figure Step 27)

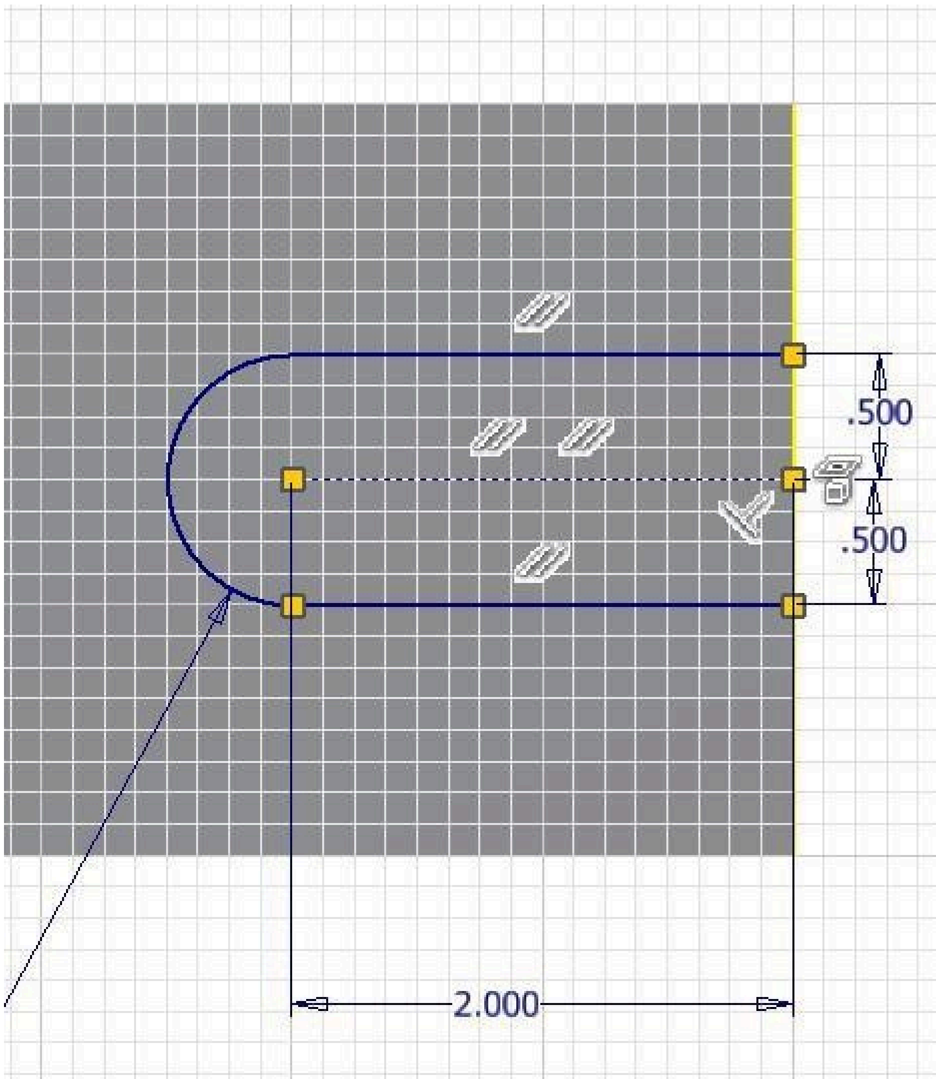


Figure Step 27

Step 28

Use what you learned in Step 14 to 17, create the additional Coincident constraints to fully constrain the sketch. (Figure Step 28)

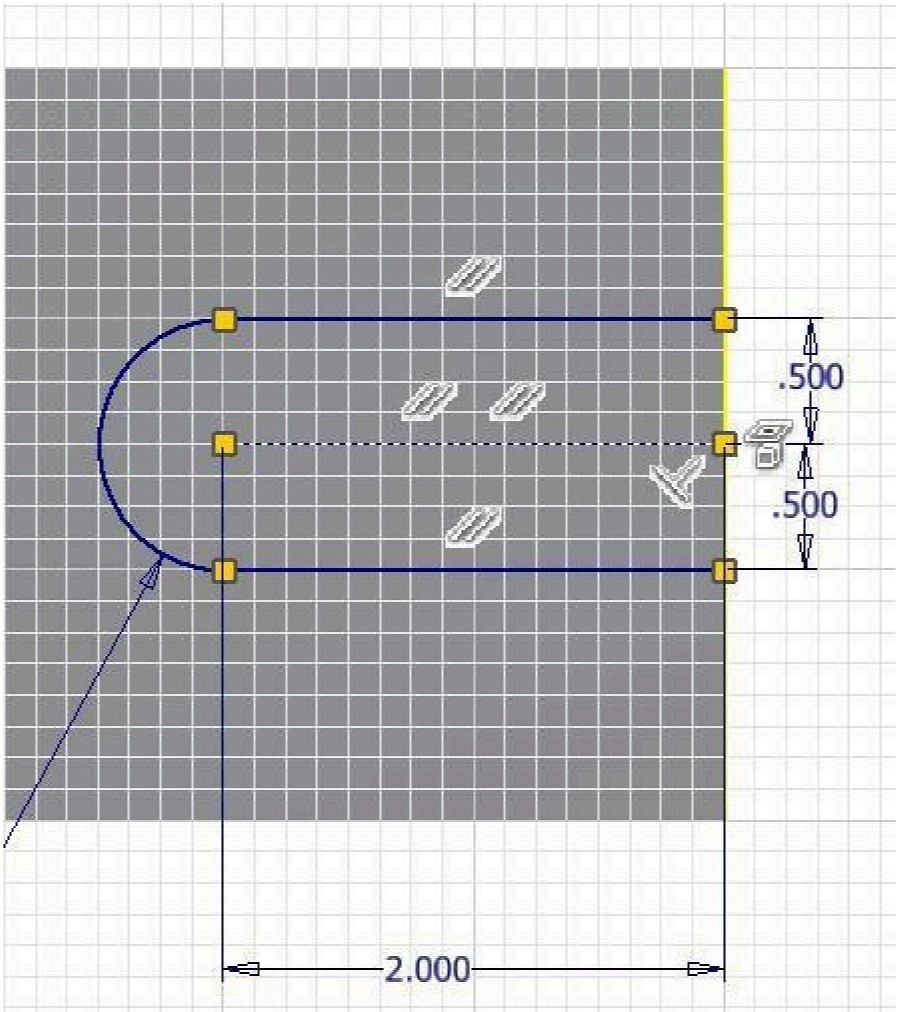


Figure Step 28

Step 29

Extrude the sketch to complete the model. (Figure Step 29)

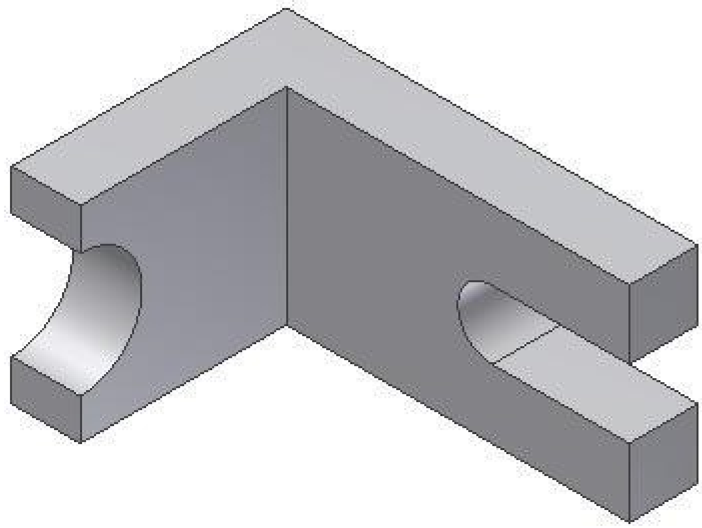


Figure Step 29

Step 30

Change the colour to: Titanium – Polished. (Figure Step 30).

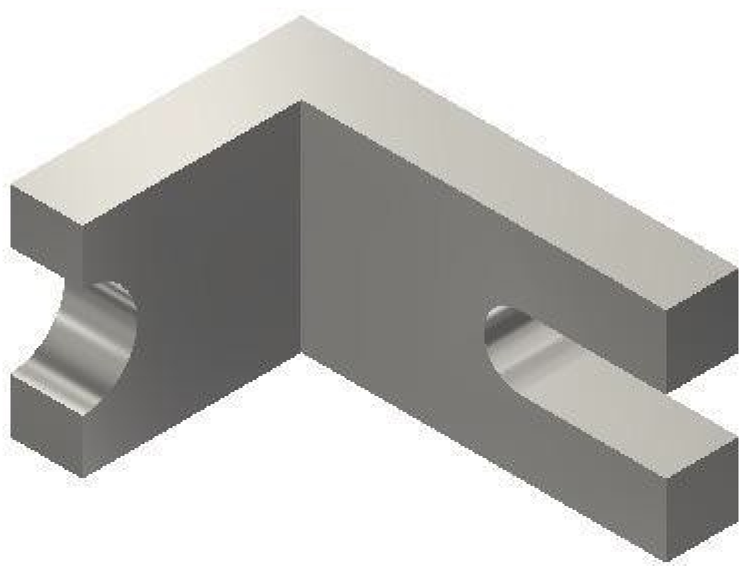


Figure Step 30

Step 31

Save and close the file.

Key Principles

Key Principles in Module 13

1. There are three commands available to draw arcs in Inventor. Similar to the CIRCLE command, the CENTER POINT ARC command also requires you to select the location of the centre point and the radius. Drawing arcs additionally requires you to locate the start point and end point or the arc. Arcs can be constructed clockwise or counterclockwise.
2. When drawing arcs, do your best to tie the centre point, the start point, and the end point to existing geometry. This will decrease the number of dimensions that are required to constrain the arc.

Lab Exercise 13-1

Time allowed: 45 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab Lab 13-1	Inventor Course	Inches	English-Modules Part (in).ipt	Chrome – Polished	N/A

Step 1

Project the Center Point onto the Base sketch.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude them to produced the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. (Figure Step 2A and 2B)

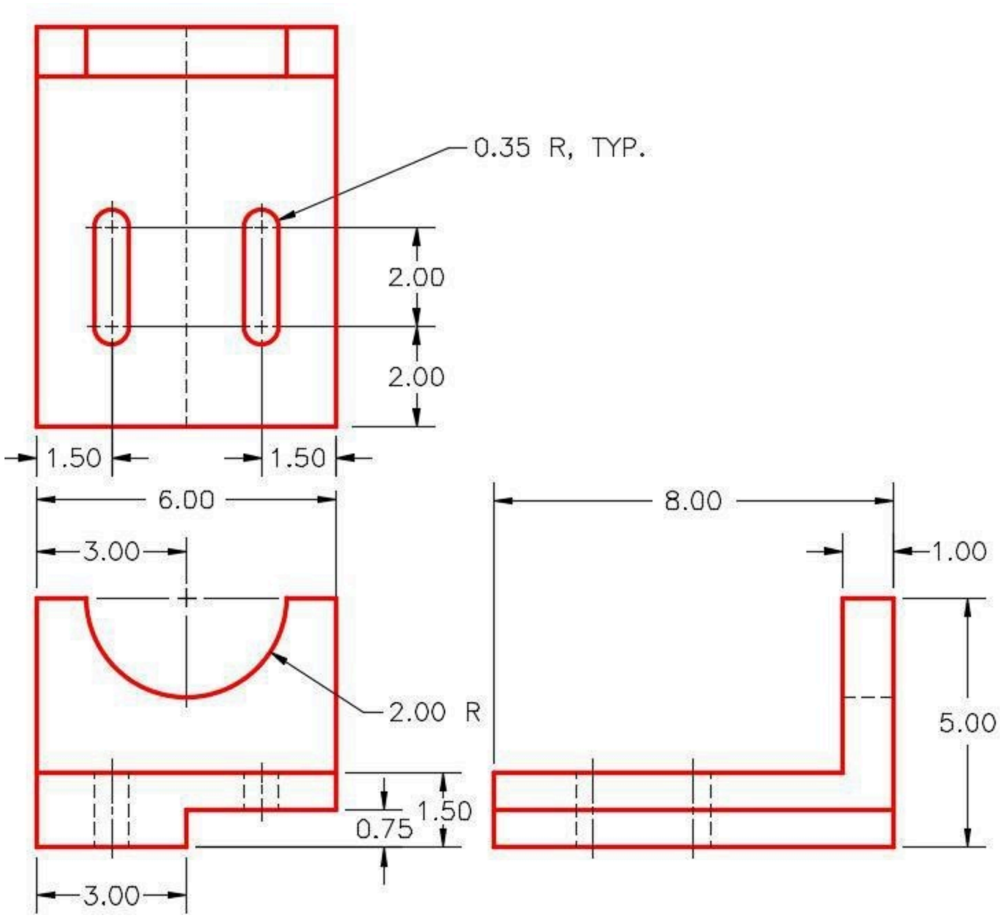


Figure Step 2A
Dimensioned Multiview Drawing [Click to see image full size]

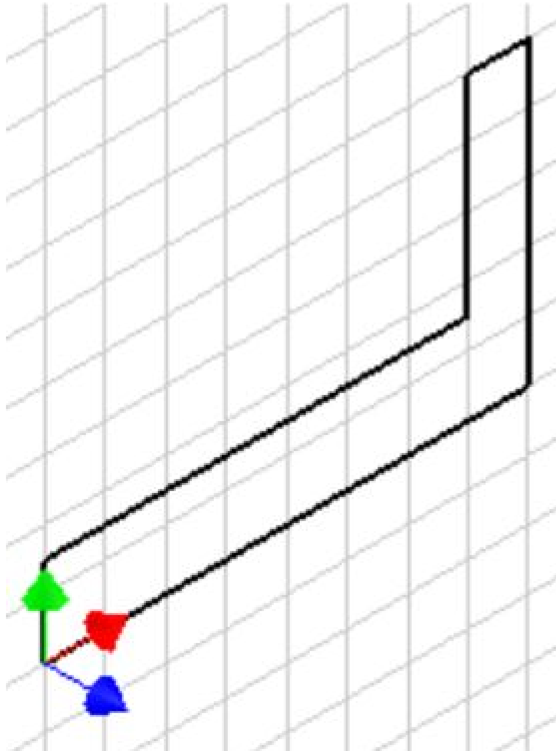


Figure Step 2B
Suggested Base Sketch
– Right Side – YZ Plane

Step 3

Apply the colour shown above. (Figure Step 3)

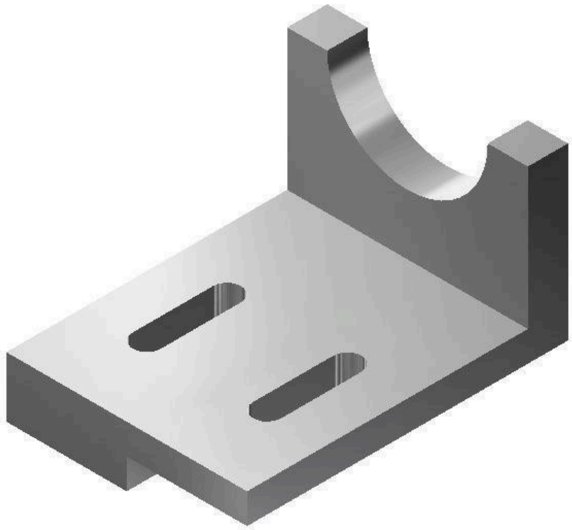


Figure Step 3 Solid Model – Home View

Lab Exercise 13-2

Time allowed: 45 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab Lab 13-2	Inventor Course	Millimeters	Metric-Modules Part (mm).ipt	Copper – Satin	N/A

Step 1

Project the Center Point onto the Base sketch.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude them to produced the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. (Figure Step 2A and 2B)

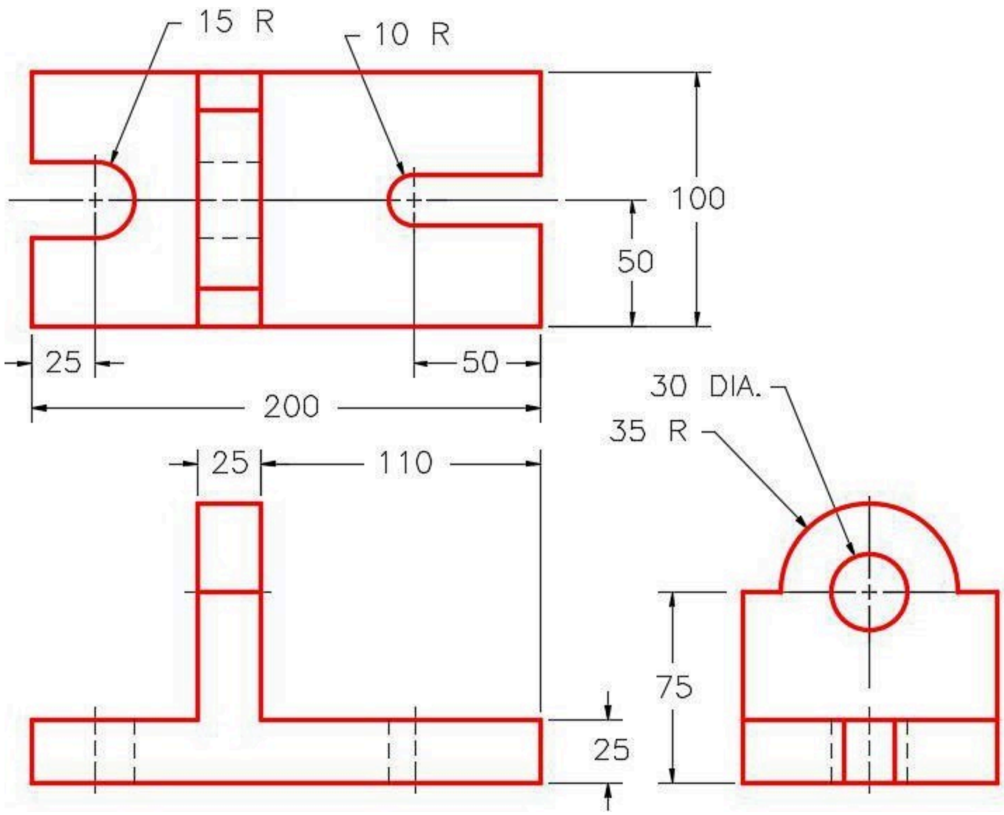


Figure Step 2A
Dimensioned Multiview Drawing [Click to see image full size]

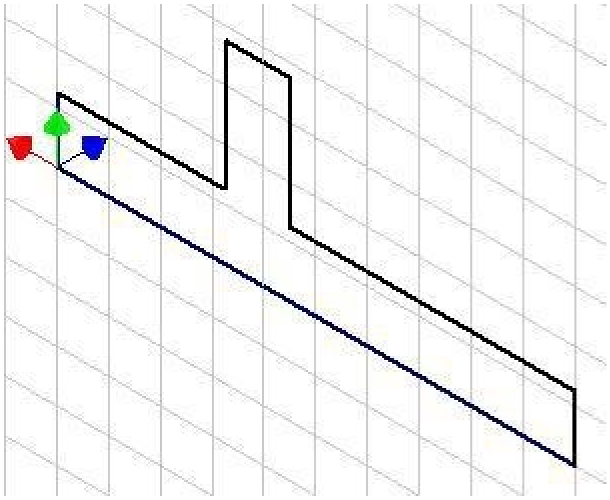


Figure Step 2B
Suggested Base Sketch –
Front – XZ Plane

Step 3

Apply the colour shown above. (Figure Step 3)

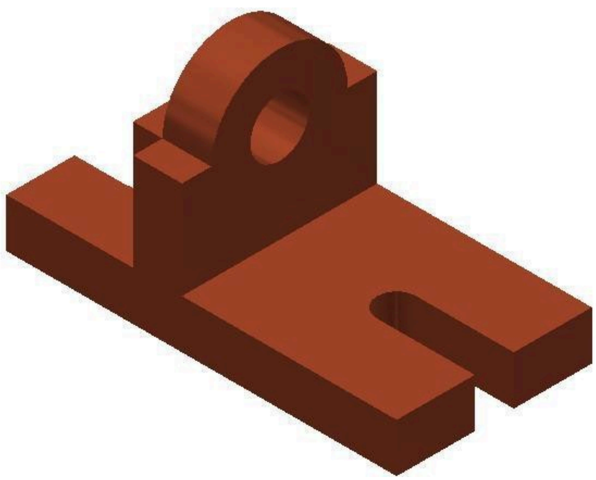


Figure Step 3
Completed Solid Model –
Home View

Module 14 Revolving

Learning Outcomes

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

1. Describe a centerline object and explain how it is inserted and used in a 2D Sketch.
2. Describe how a Base sketch is revolved with and without the use of a centerline to create a solid model.
3. Apply the REVOLVE command to create a solid model from a Base sketch.

Revolving

When drawing symmetrical objects it is much easier to create the model by *revolving* the Base sketch around an *axis* rather then extruding it. The axis, which can be one of the lines in the sketch or a centerline, must always be located in the centre of the symmetrical model. The sketch can be revolved any angle between 0 and 360 degrees.

In this module, the basic features of the REVOLVE command are taught. The Inventor Advanced book will cover the more advanced features.

The models in Figure 14-1 and 14-2 were created by revolving the same Base sketch around an axis. Take note how the two solid models that were created using the same sketch are quite different. In Figure 14-1, the line on left side of the sketch was used as the axis while in Figure 14-2, it was the centerline that was used as an axis or revolution.

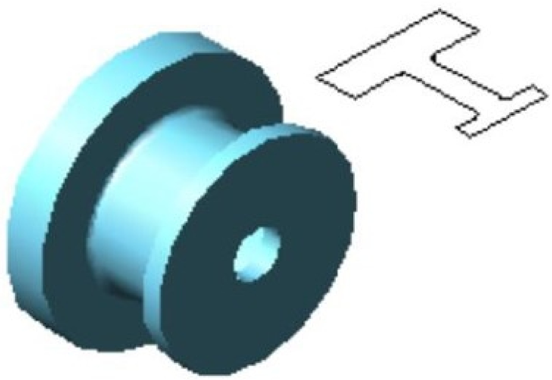


Figure 14-1
A 2D Sketch Revolved Around a Line in the Sketch

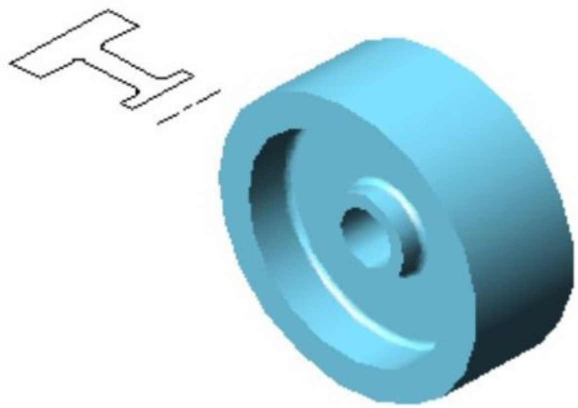
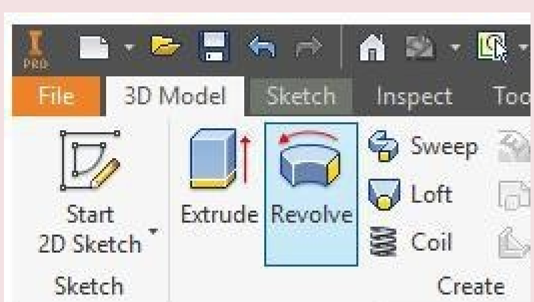


Figure 14-2
The Same 2D Sketch Revolved around a Centerline

Inventor Command: REVOLVE

The REVOLVE command is used to create a solid model by revolving the Base sketch around an axis.

Shortcut: **R**



Centerlines

A *centerline* is a line with its properties set to act as a centerline. In the REVOLVE command, a centerline is automatically recognized as the axis for the revolution. The two methods of drawing a centerline, which is similar to drawing construction a line, are as follows:

Method 1

Draw the line using the LINE command and then select it. While it is selected, click the Centerline icon.

Method 2

Enable the Centerline icon and then draw the line, using the LINE command. The Centerline icon is shown in Figure 14-3. A centerline will display as the centerline linetype.

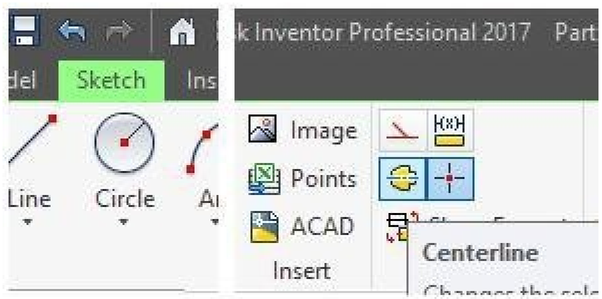


Figure 14-3
Centerline Icon

WORK ALONG: Revolving a Sketch Without using a Centerline

Step 1

Check the default project and if necessary, set it to Inventor Course.

Step 2

Using the NEW command, start a new part file using the template: English-Modules Part (in).ipt.

Step 3

Save the file with the name: Inventor Workalong 14-1. (Figure Step 3A and 3B)

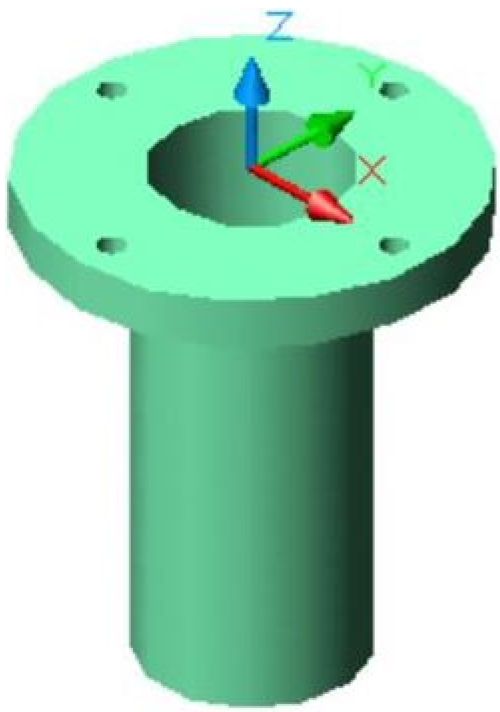


Figure Step 3A
3D Model

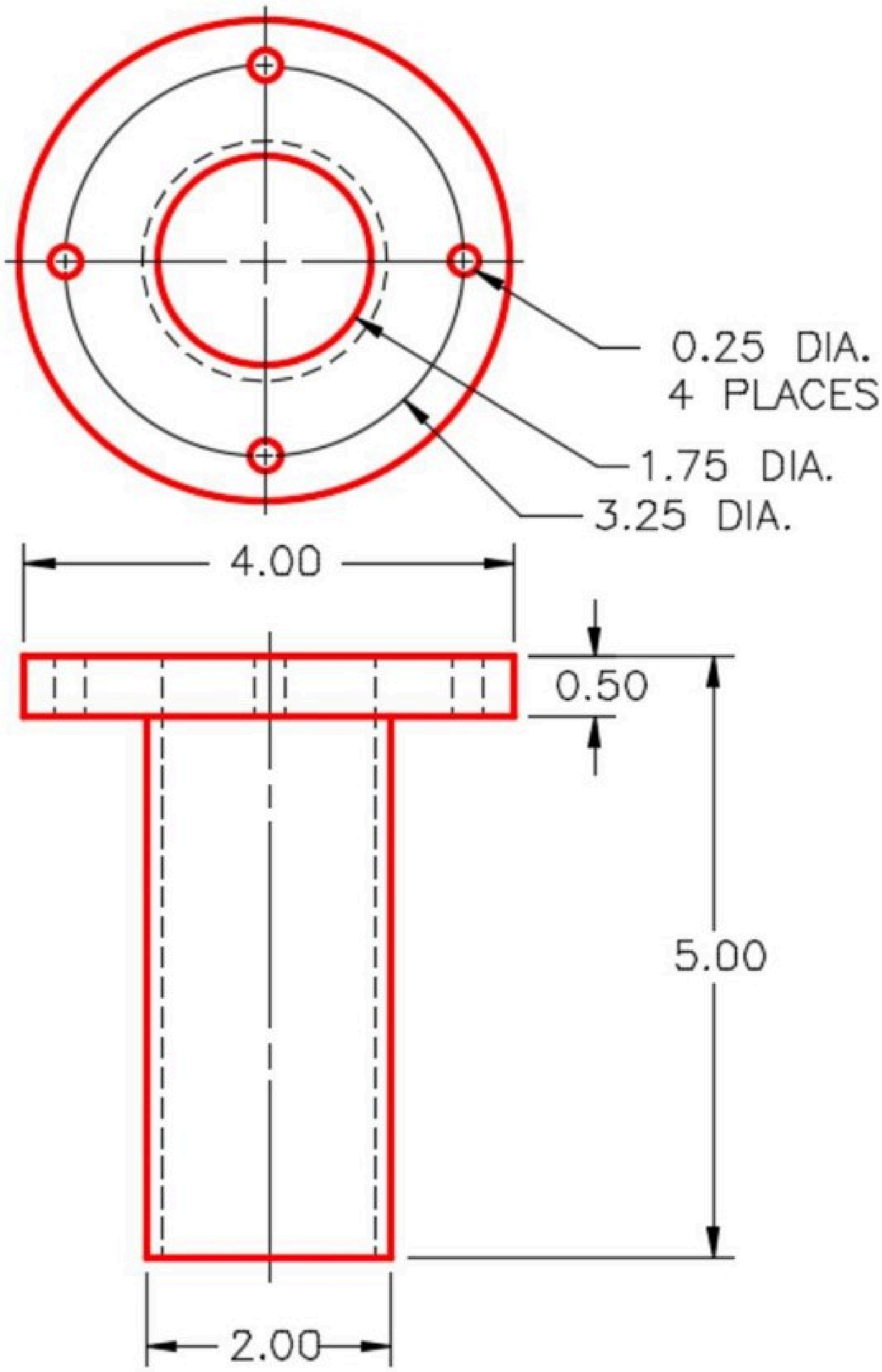


Figure Step 3B
Dimensioned Multiview Drawing [Click to see image full size]

AUTHOR'S COMMENTS: In this and the next workalong, you will be constructing the same solid model. In this workalong without the use of a centerline, and the next workalong with the use of a centerline. Either method is an acceptable way of creating the solid model. These two workalongs will allow you to practice both methods.

Step 4

Start a new sketch on the Front or XZ Plane. Project the Center Point onto the sketch.

Step 5

Draw and dimension one-half of the Front view as shown in the figure. Ensure that the sketch is fully constrained (Figure Step 5)

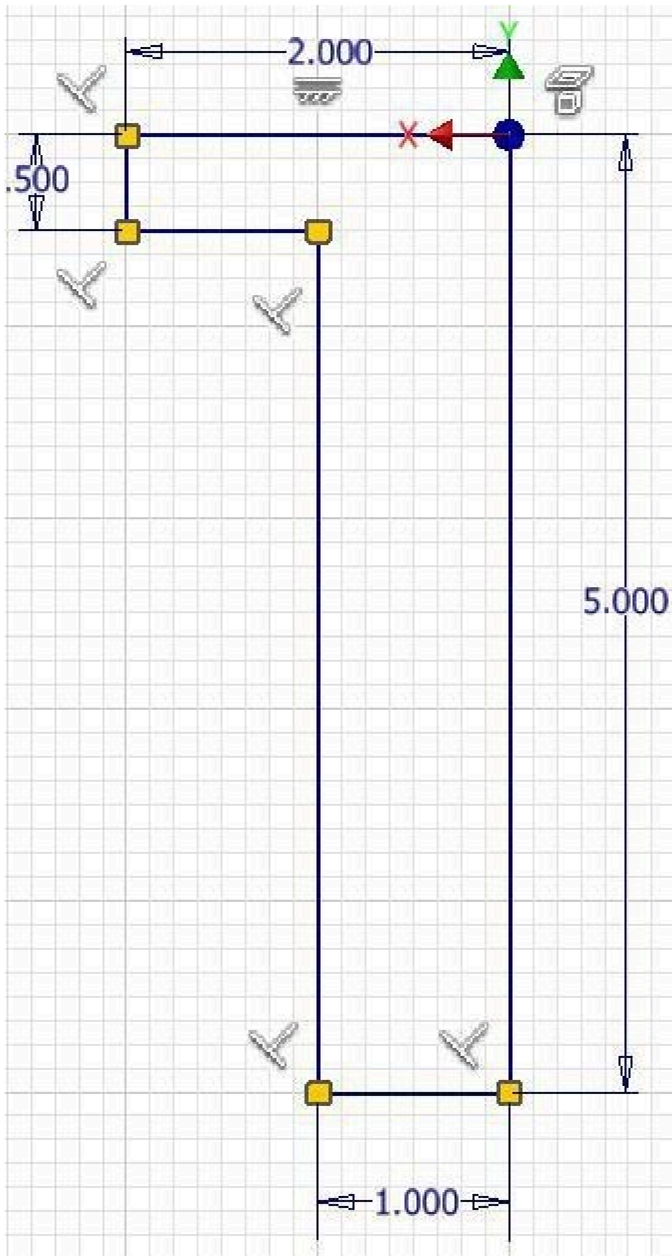


Figure Step 5

AUTHOR'S COMMENTS: Note how the sketch is a profile of the object, that when revolved, will create the solid model. For now, ignore the centre hole and the 4 small holes. The line located in the centre of the model is used as the axis. The holes will be inserting after the sketch is revolved to complete the solid model.

AUTHOR'S COMMENTS: Your geometrical and dimensional constrains may not match the figure exactly. Ensure that the sketch is fully constrained.

Step 6

In Model mode, enter the REVOLVE command. It will highlight the sketch automatically as the area to revolve. (Figure Step 6)

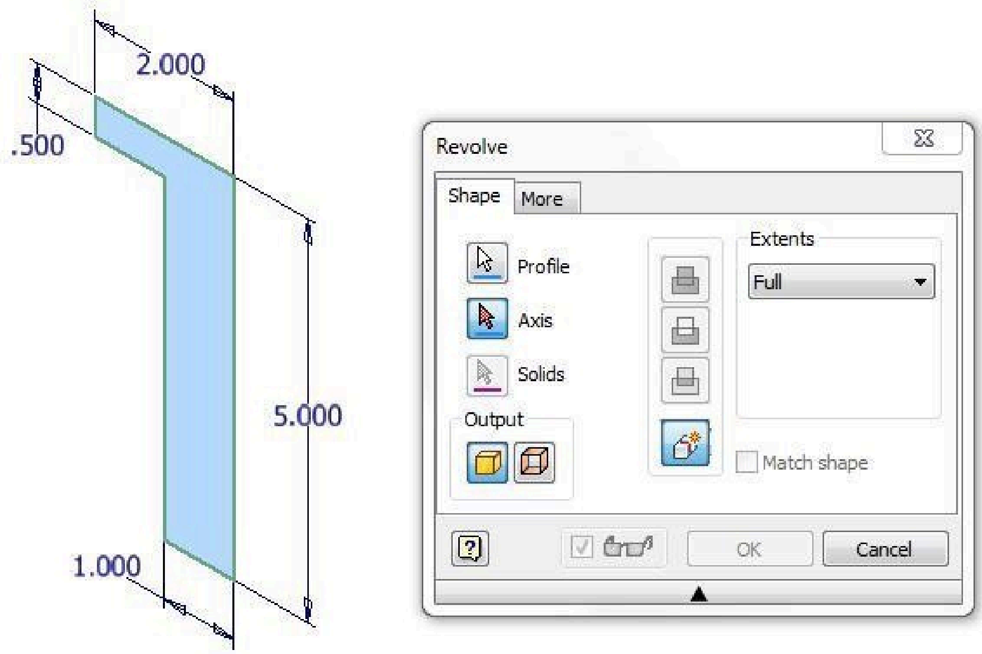


Figure Step 6 [Click to see image full size]

Step 7

In the Revolve dialogue box, set the Extents to Full and enable the Axis icon. Select the line on the right side of the sketch as the axis. The Full setting means that it will be revolved 360 degrees. (Figure Step 7)

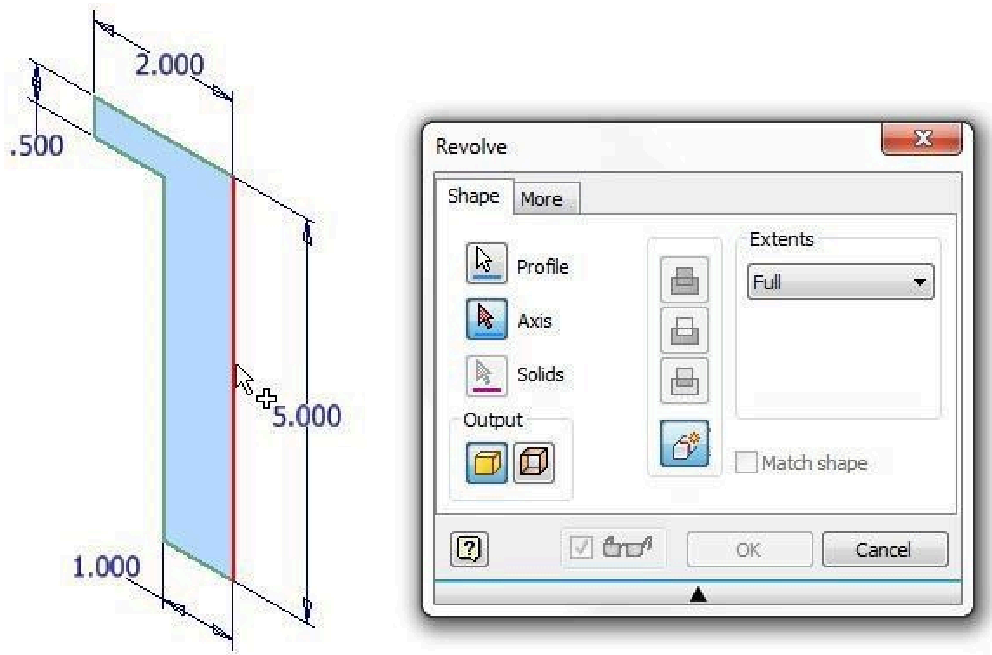


Figure Step 7 [Click to see image full size]

Step 8

After you select the axis, the REVOLVE command will display the Base model as it is revolved. If this is the desired outcome, click OK. (Figure Step 8A and 8B)

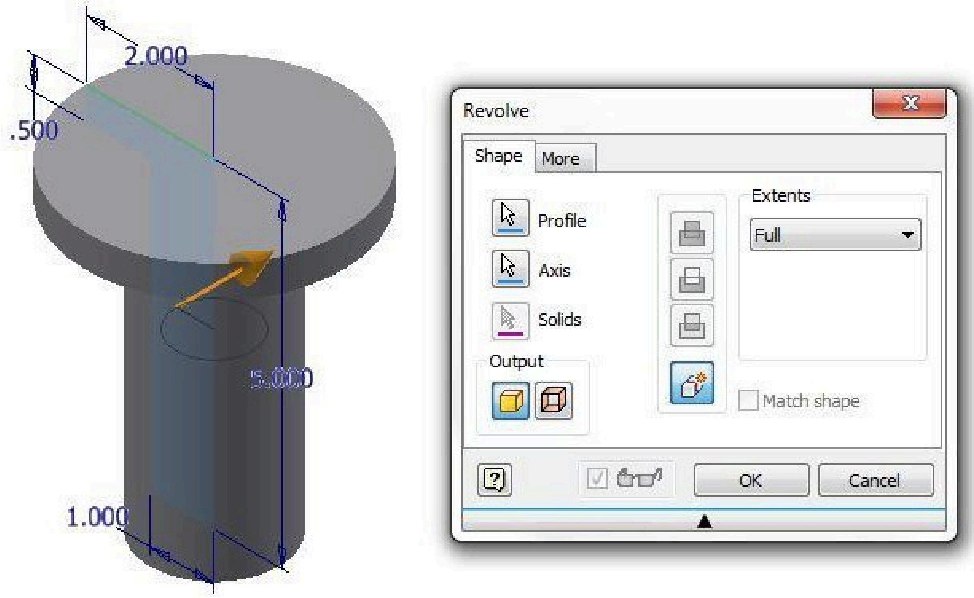


Figure Step 8A [Click to see image full size]

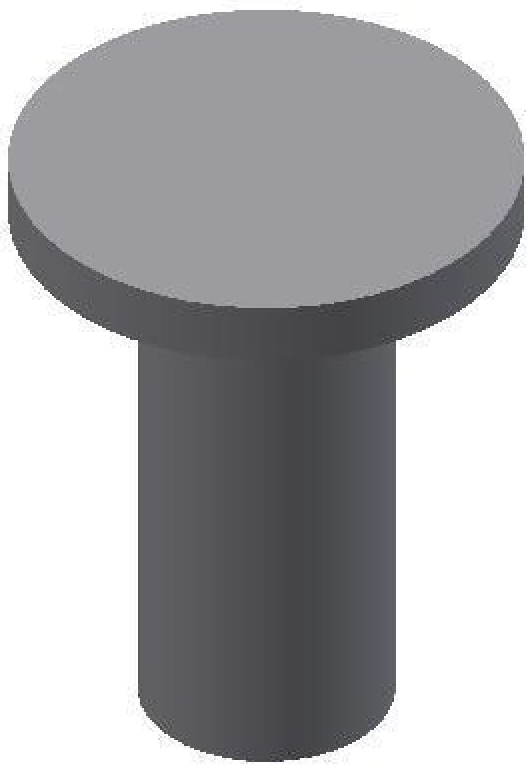


Figure Step 8B

Step 9

Start a new sketch on the top plane of the model. (Figure Step 9)

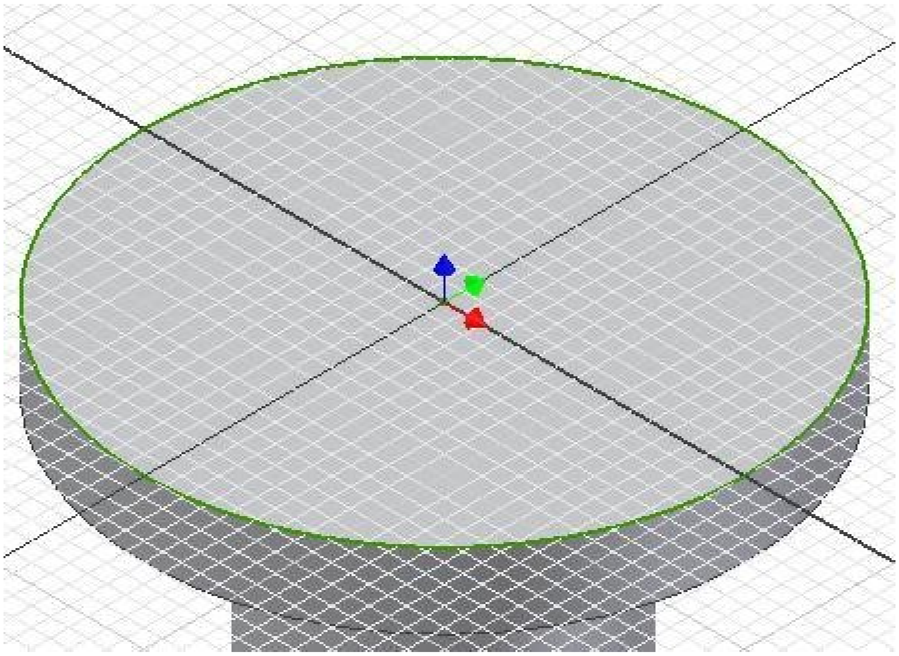


Figure Step 9

Step 10

Using what you learned in Module 12, draw a construction circle and four construction lines. Insert a dimension for the diameter of the circle. Ensure that the sketch is fully constrained. (Figure Step 10)

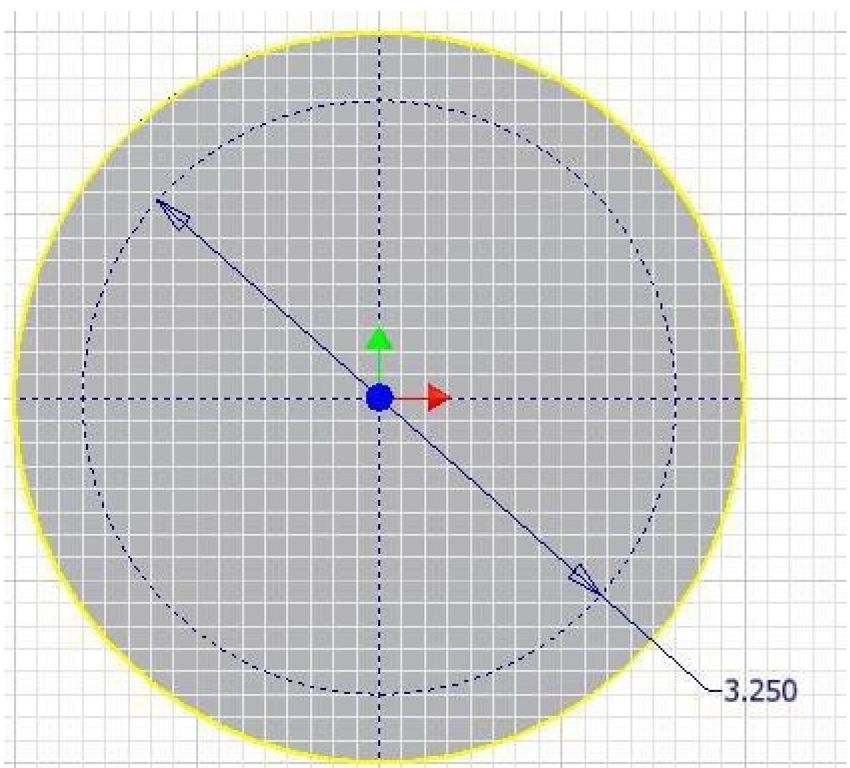


Figure Step 10

AUTHOR'S COMMENTS: Ensure that you snap the lines correctly when you draw them. If you don't, you will have trouble fully constraining the sketch. If you have trouble doing this step, look at the workalong in Module12.

Step 11

Using the technique that you learned in Module 12, draw the 4 circles. Dimension one and constrain the additional 3 with an Equal constraint. (Figure Step 11A and 11B)

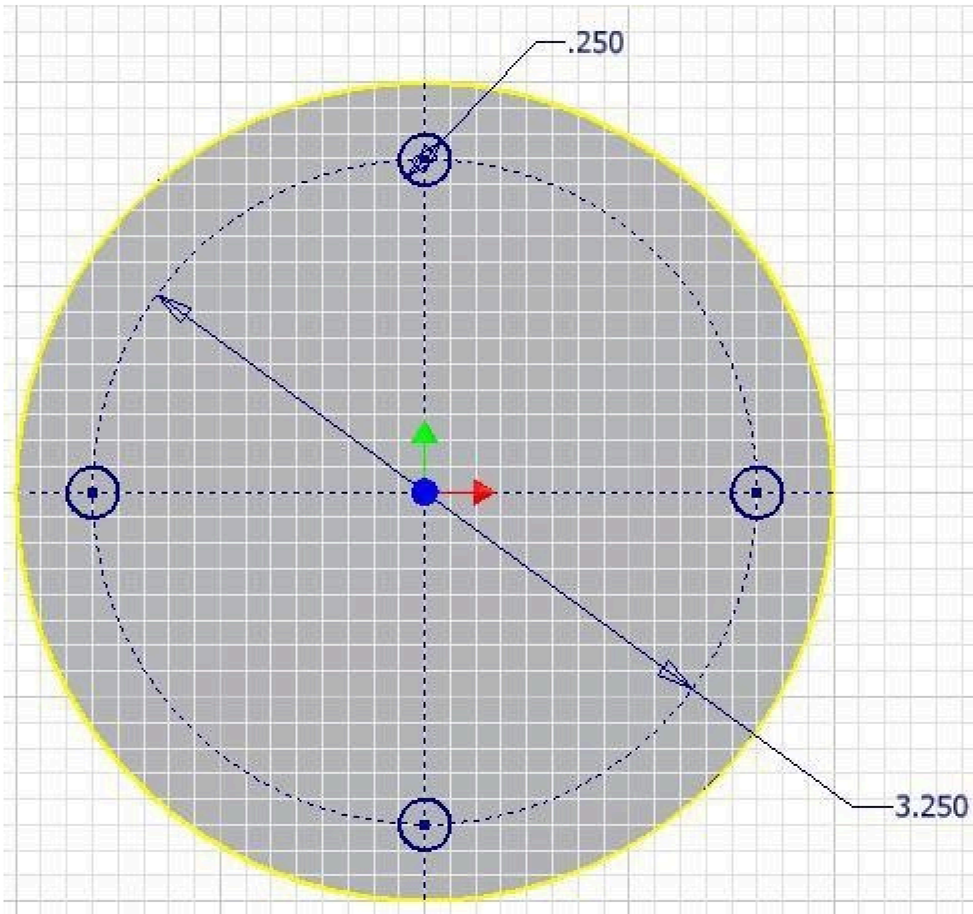


Figure Step 11A

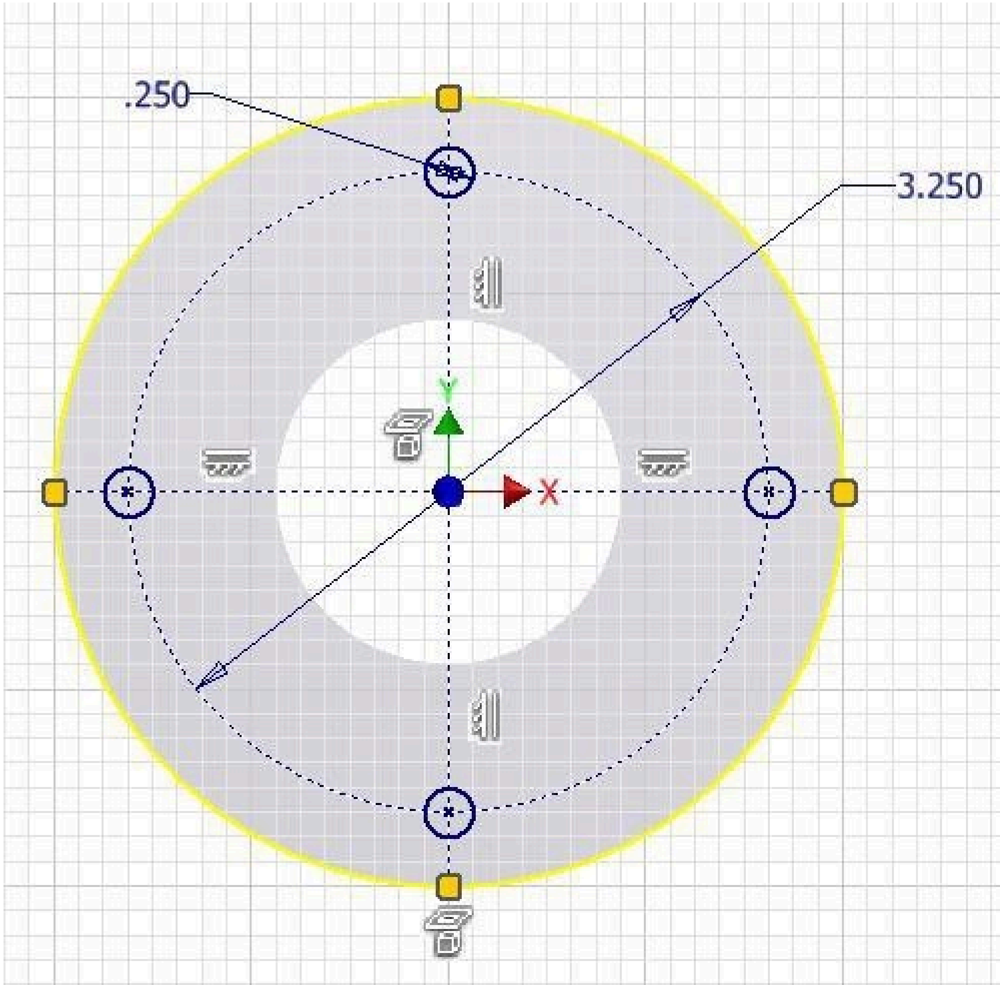


Figure Step 11B

AUTHOR'S COMMENTS: Your geometrical and dimensional constrains may not match the figure exactly. Ensure that your sketch is fully constrained.

Step 12

Extrude the four circles to the To Next extents. (Figure Step 12)

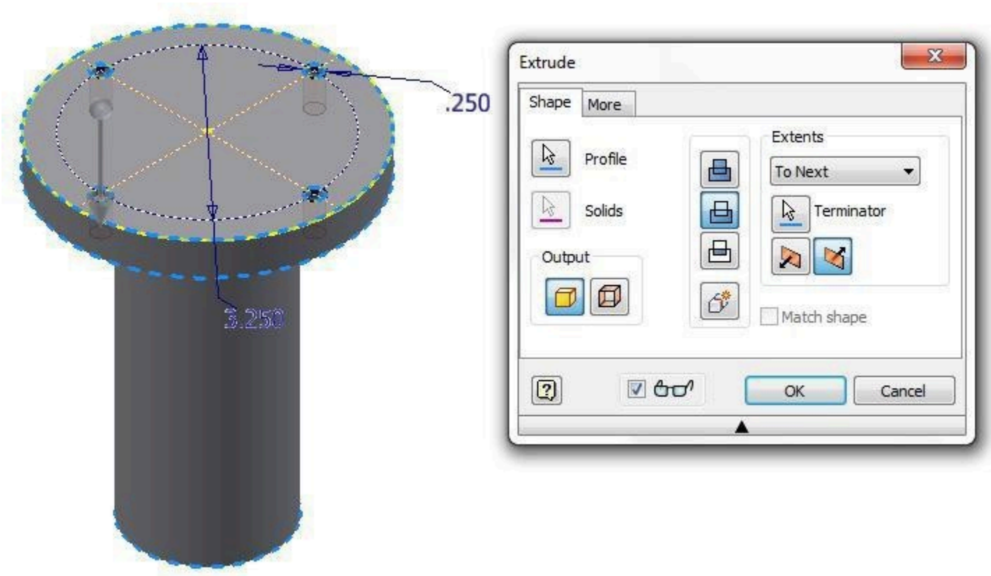


Figure Step 12 [Click to see image full size]

Step 13

Start another sketch on the top plane and draw a circle by offsetting the outside diameter. Dimension the circle and extrude it to complete the model. (Figure Step 13)

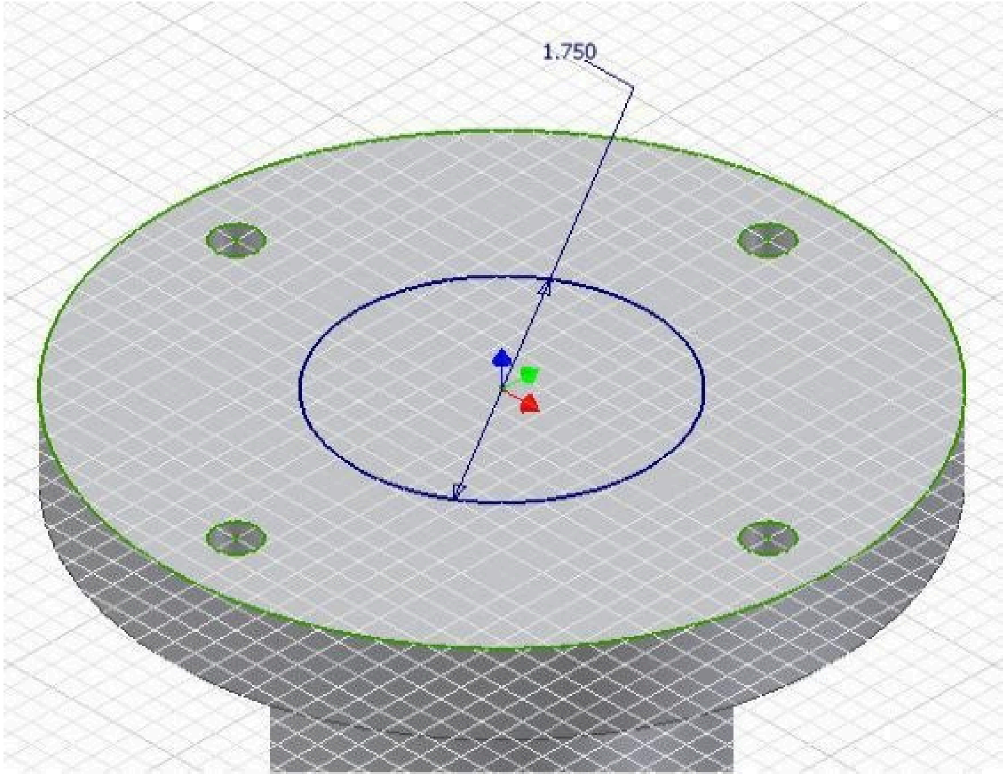


Figure Step 13 [Click to see image full size]

AUTHOR'S COMMENTS: Using an offset, will automatically constrain the circle.

Step 14

Change the colour to: Aluminum – Polished. (Figure Step 14)

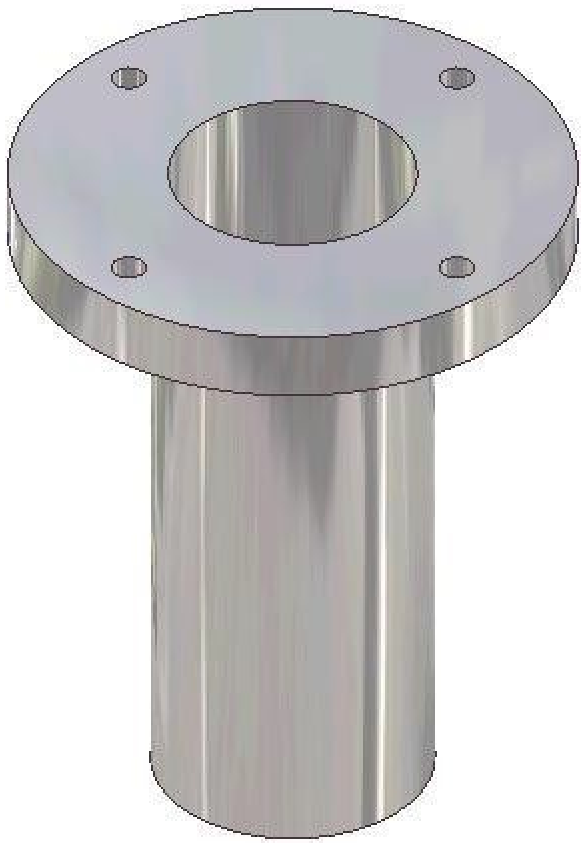


Figure Step 14

Step 15

Save and close the file.

WORK ALONG: Revolving a Sketch using a Centerline

Step 1

Check the default project and if necessary, set it to Inventor Course.

Step 2

Enter the NEW command to start a new part file using the template: English-Modules Part (in).ipt.

Step 3

Save the file with the name: Inventor Workalong 14-2. (Figure Step 3)

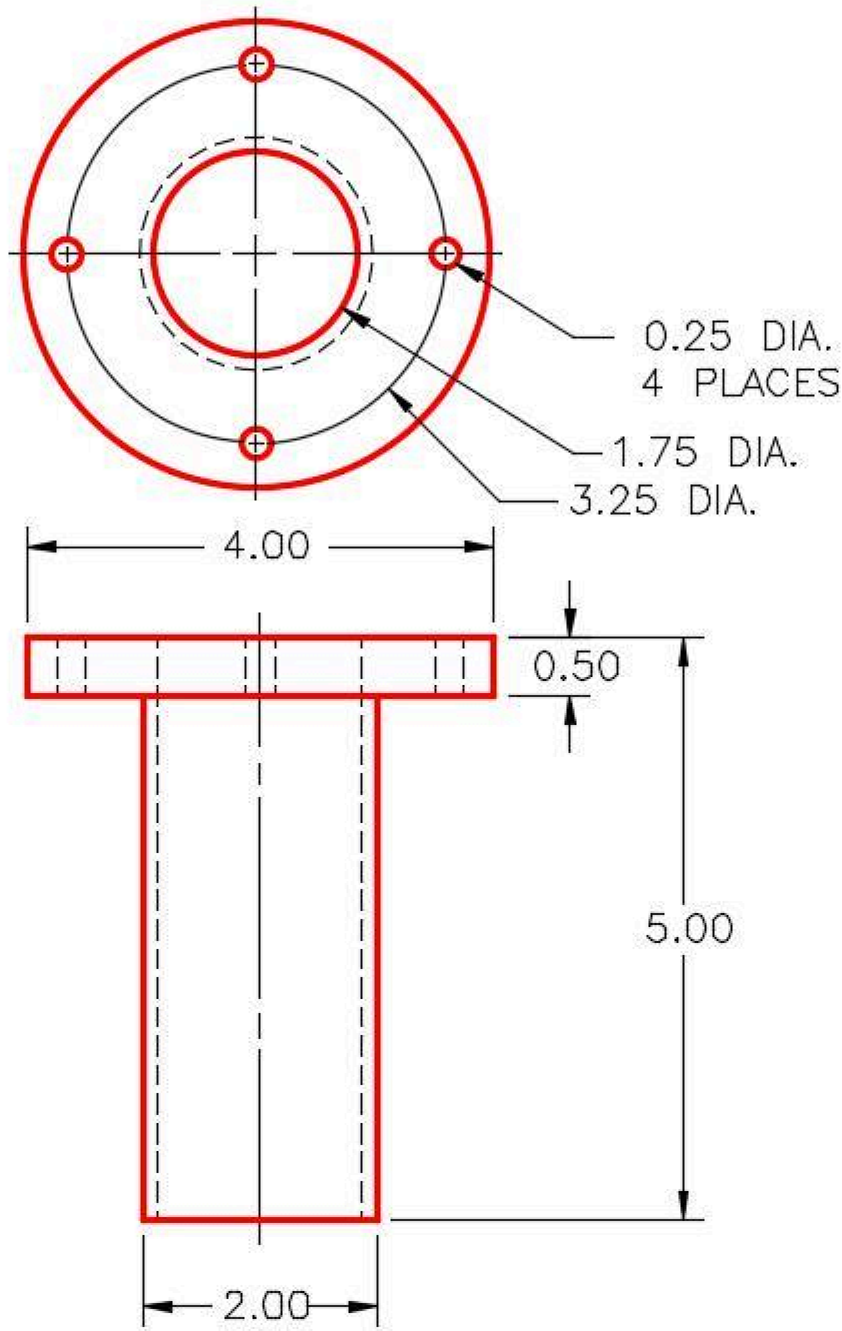


Figure Step 3
Dimensioned Multiview Drawing

Step 4

Start a new sketch on the Front or XZ Plane. Draw and dimension a line start it by snapping to the Center Point. Draw it 5 inches in the negative Y direction. This is the length of the model. This is centerline of the solid model. Change the line's properties to a centerline. (Figure Step 4A and 4B)

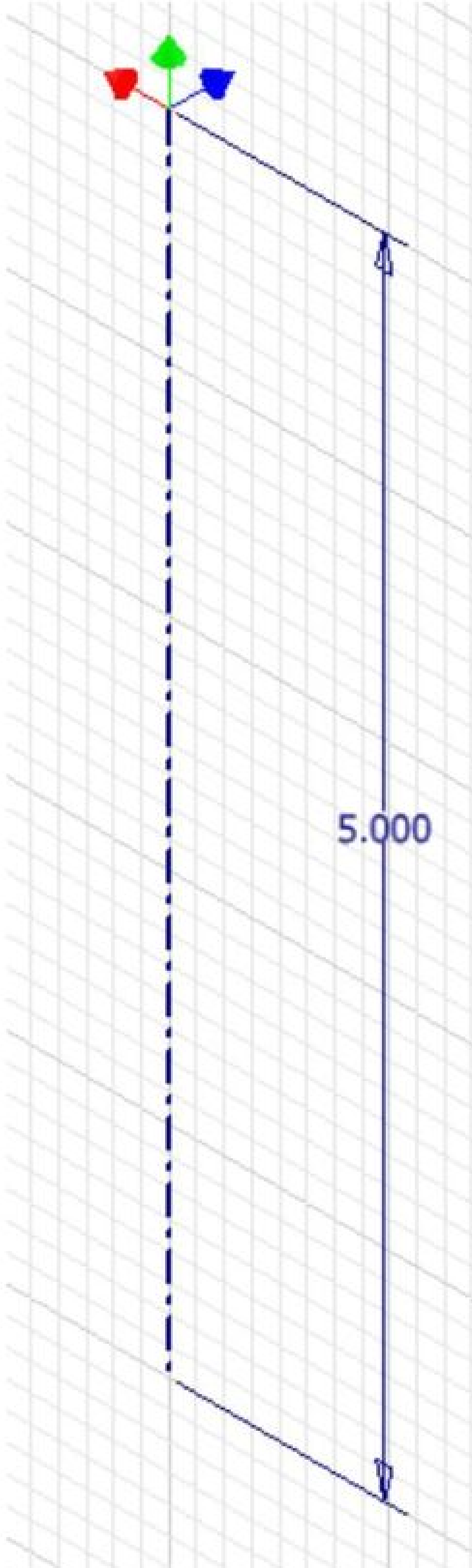


Figure Step 4A

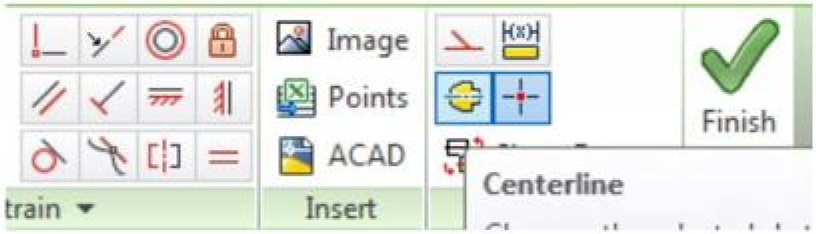


Figure Step 4B

Step 5

Draw and dimension one-half of the Front view as shown in the figure. (Figure Step 5A and 5B)

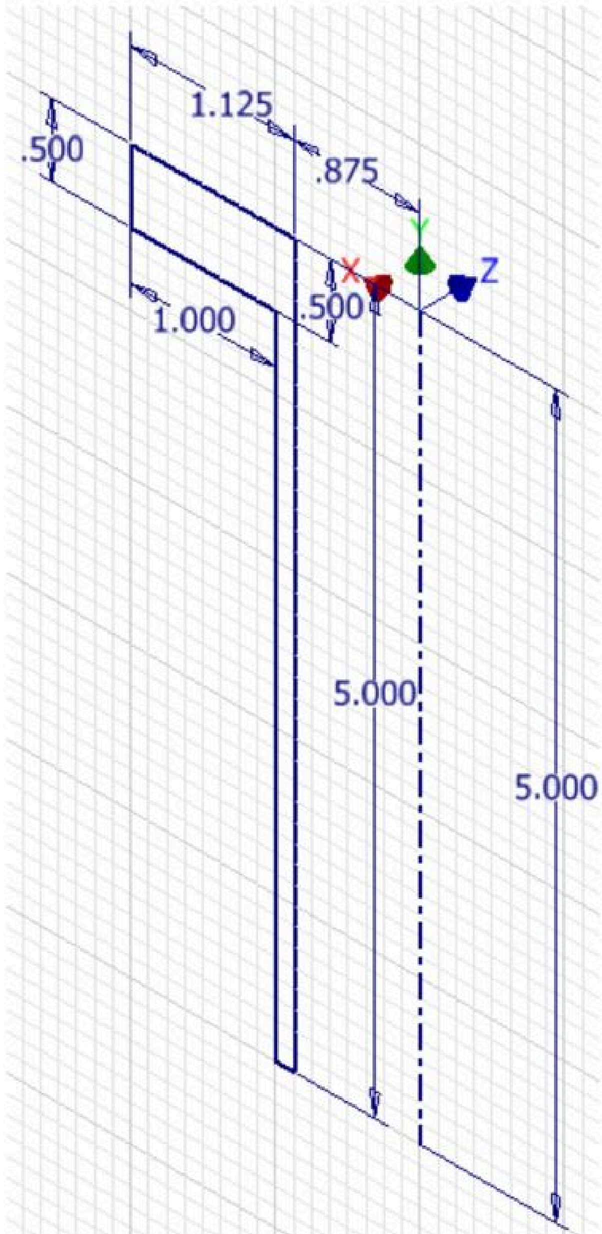


Figure Step 5A [Click to see image full size]

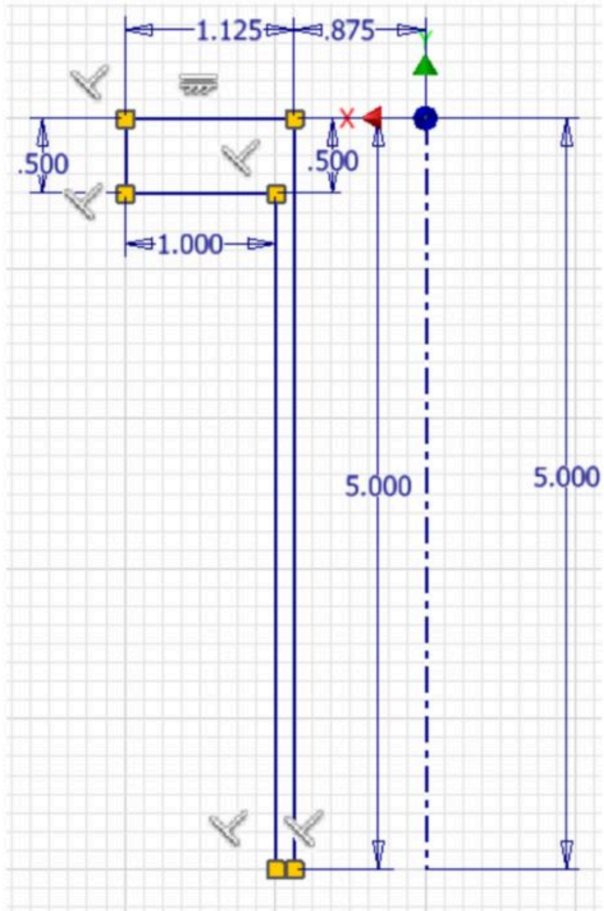


Figure Step 5B [Click to see image full size]

AUTHOR'S COMMENTS: Your geometrical and dimensional constrains may not match the figure exactly. Just ensure that your sketch is fully constrained.

Step 6

Return to Model mode and enter the REVOLVE command. Since a centerline is part of the sketch, the REVOLVE command will automatically use it as the axis to revolve the sketch around. It will display the outcome of the revolution. (Figure Step 6)

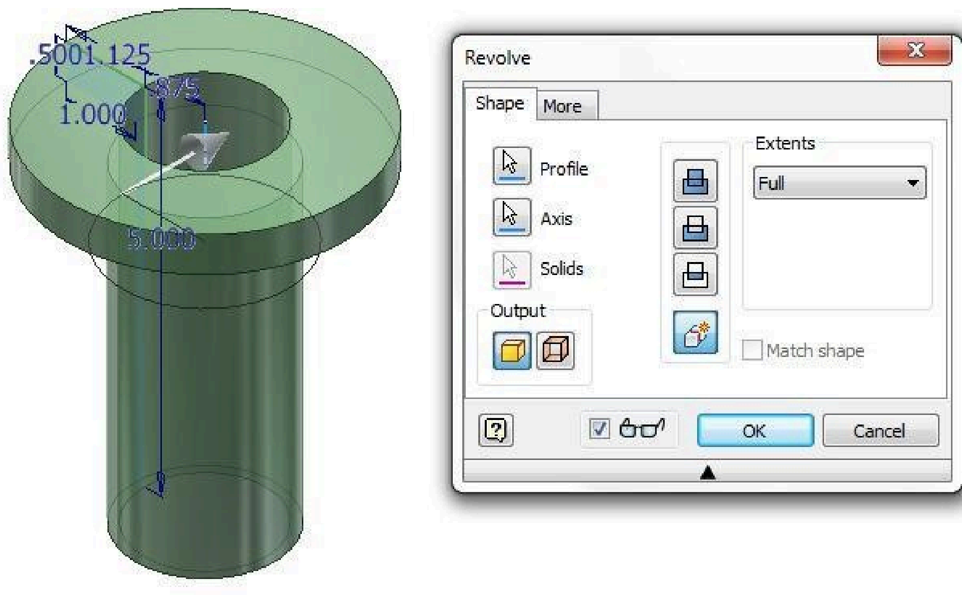


Figure Step 6 [Click to see image full size]

Step 7

In a new sketch, add the four smaller circles and extrude them to complete the model.

Step 8

Change the colour to: Aluminum – Polished. (Figure Step 8)

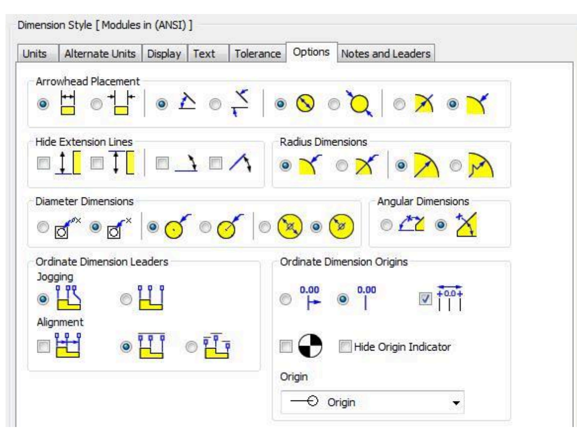


Figure Step 8

Step 9

Save and close the file.

Drafting Lesson: Cross Sections

When a view of an object requires a clearer description of its interior or it is hard to dimension because of the hidden lines, a cross section view can be drawn in place of the normal multiview. See Figure 32-1

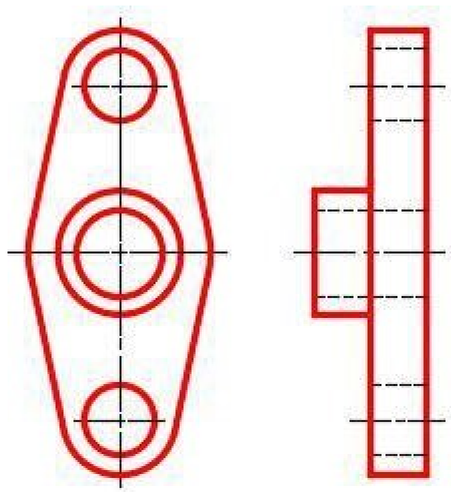


Figure 32-1
Normal Front and Right Side View of an Object

A *cross section* view, also called a *section*, is a view of the object as if it were cut along a cutting plane and the two pieces pulled apart exposing the inside of the object. See Figure 32-2 and 32-3.

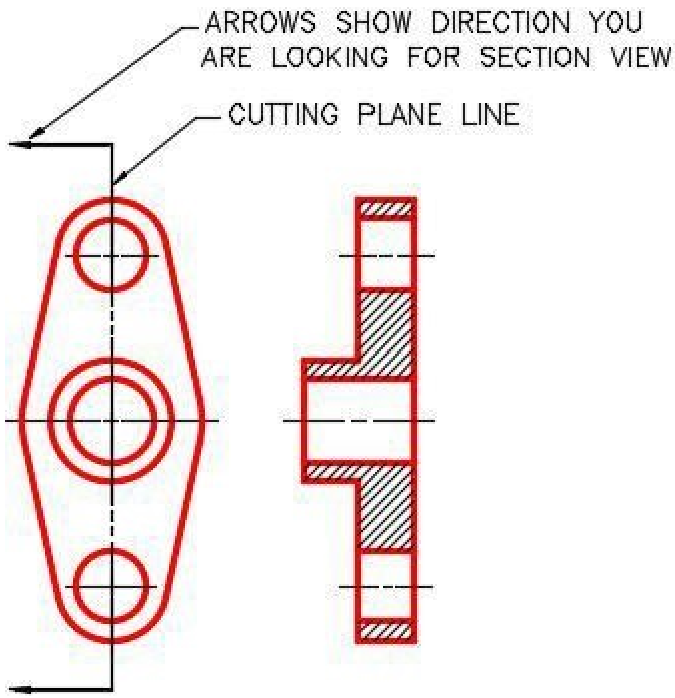


Figure 32-2
A Front view and Right
Side Section View

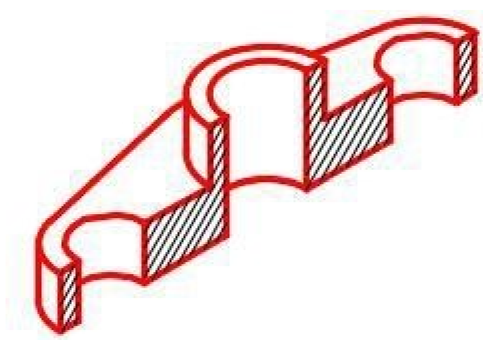


Figure 32-3
3D Model of the Object
Showing it Cut on the
Cutting Plane Line

A *cutting plane* line is the line along the object where the cut would have been made. See Figure 32-2. The arrows point in the direction that you are looking when drawing the section view. The surfaces of the object that are solid, when cut, are crosshatched.

Key Principles

Key Principles in Module 14

1. When drawing symmetrical objects, it is much easier to create the model by revolving the Base sketch around an axis rather then extruding it. The axis, which can be one of the lines in the sketch or a centerline, must always be located in the centre of the symmetrical MODEL.
2. A centerline is a line with its properties set to act as a centerline.

Lab Exercise 14-1

Time allowed: 60 minutes.

Part Name	Project	Units	Template	Color	M
Inventor Lab Lab 14-1	Inventor Course	Millimeters	Metric-Modules Part (mm).ipt	Copper – Polished	N/

Step 1

Project the Center Point onto the Base plane.

Step 2

Note the location of X0Y0Z0. Draw the Base sketch and revolve it create the Base model of the object shown below. Revolve it by using a line in the sketch. Do not draw a centerline. (Figure Step 2A and 2B)

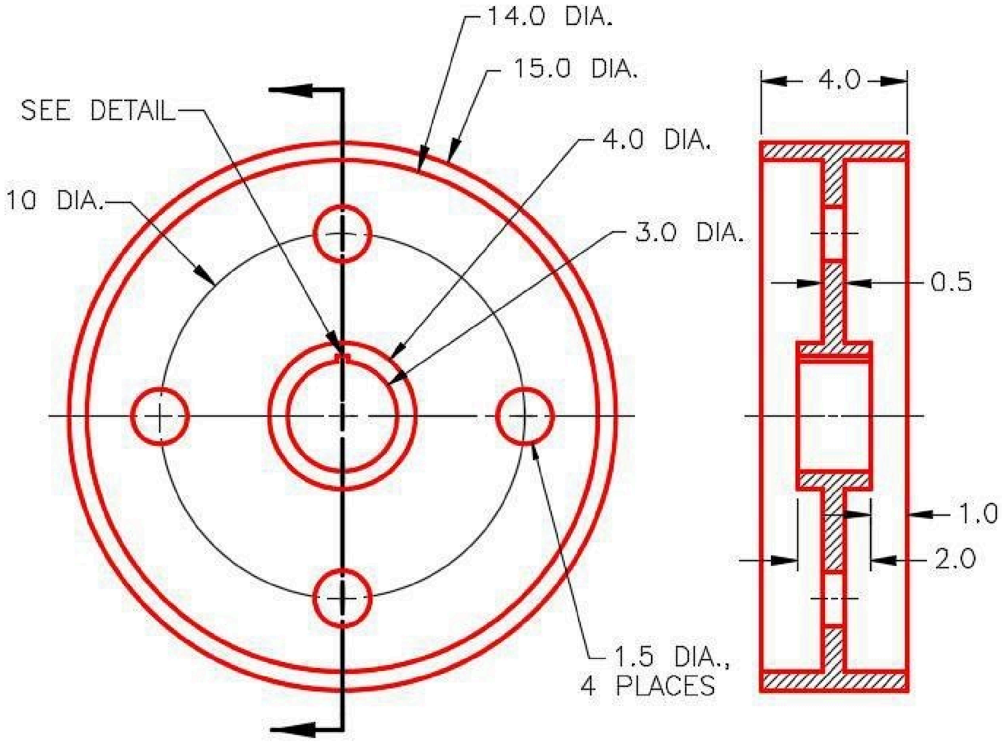


Figure Step 2A
Dimensioned Multiview Drawing

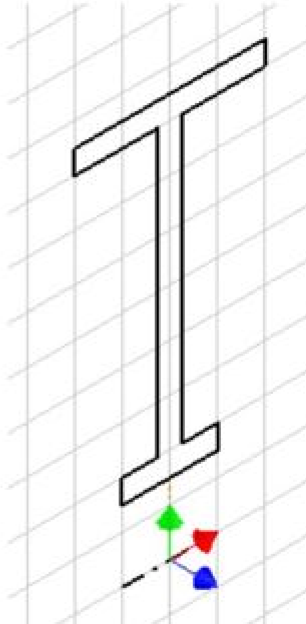


Figure Step 2B Suggested Base Sketch – Right Side – YZ Plane

Step 3

Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches.

Step 4

Apply the colour shown above. (Figure Step 4A and 4B)

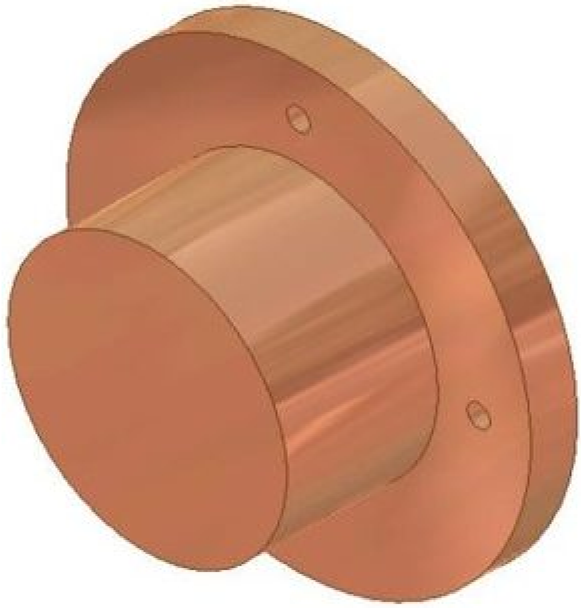


Figure Step 4A
Completed Solid Model
– Home View

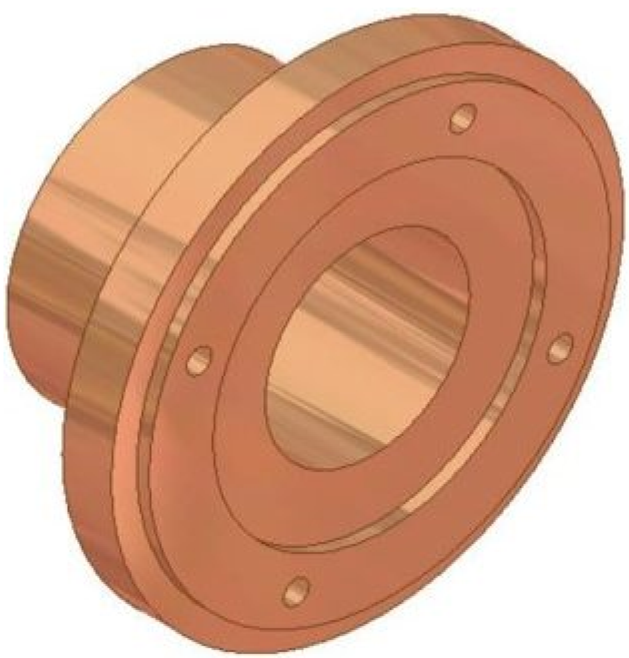


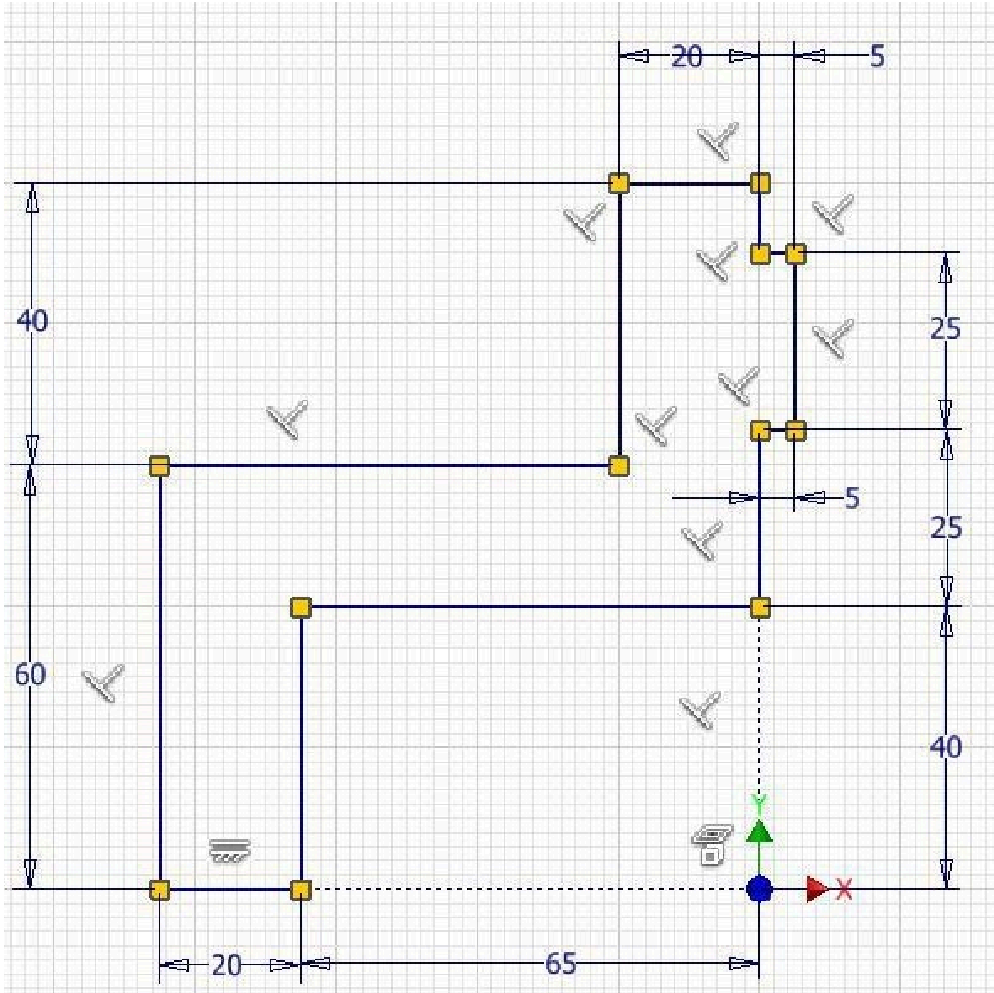
Figure Step 4B
Completed Model
– Rear View

Step 5

Add the four small holes on a new sketch and extrude them.

AUTHOR'S GEOMETRIC CONSTRAINS: The following figures shows the sketch's construction method plus geometric and dimensional constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketch using a different construction method and constraints, that is what is important. You may want to compare your construction method and constraints with the authors.

AUTHOR'S COMMENTS: Your geometrical and dimensional constrains may not match the figure exactly. Just ensure that your sketch is fully constrained.



Base Sketch

Lab Exercise 14-2

Time allowed: 60 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab Lab 14-2	Inventor Course	Inches	English-Modules Part (in).ipt	Zinc	N/A

Step 1

Project the Center Point onto the Base plane.

Step 2

Note the location of X0Y0Z0. Draw the Base sketch and revolve it create the Base model of the object shown below. Revolve it by using a centerline. (Figure Step 2A, 2B, 2C, 2D, and 2E)

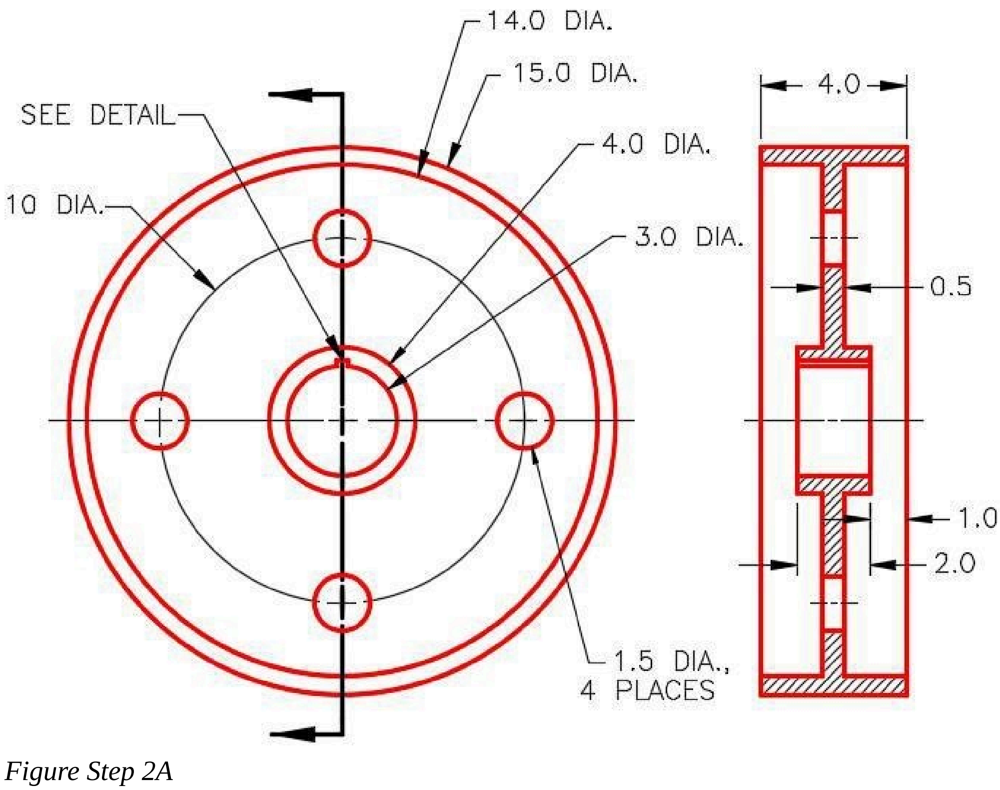


Figure Step 2A
Dimensioned Multiview Drawing

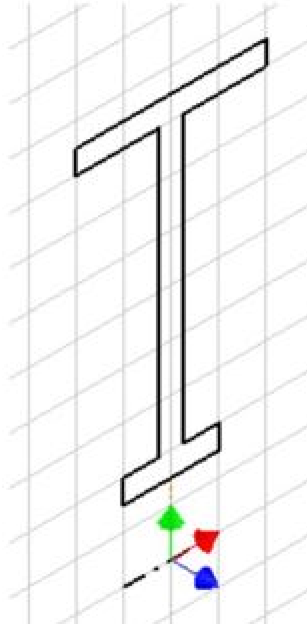


Figure Step 2B
Suggested
Base Sketch –
YZ Plane

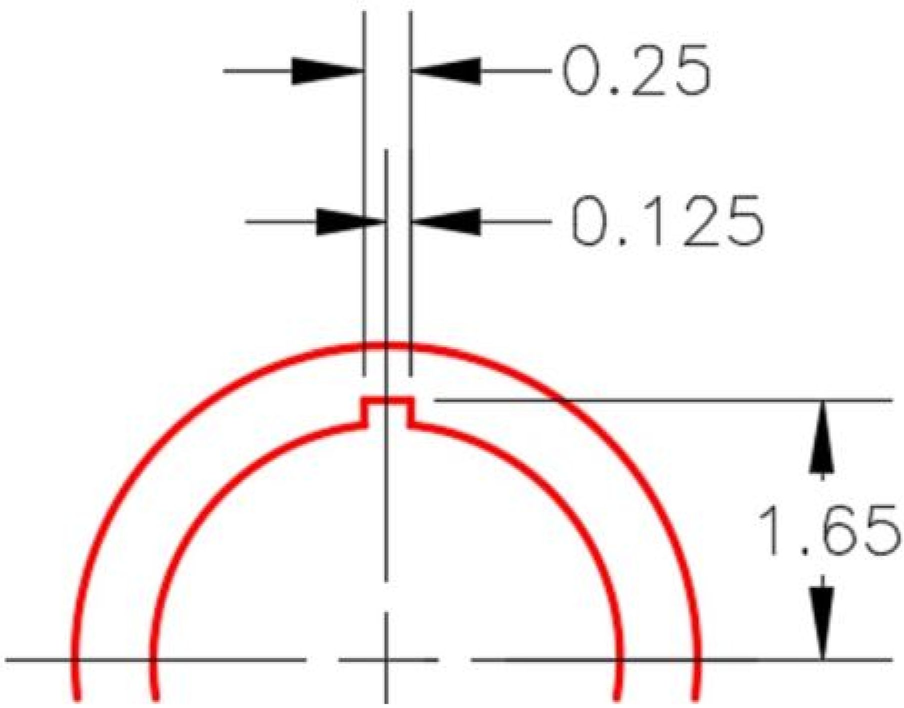


Figure Step 2C
Detail of Keyway

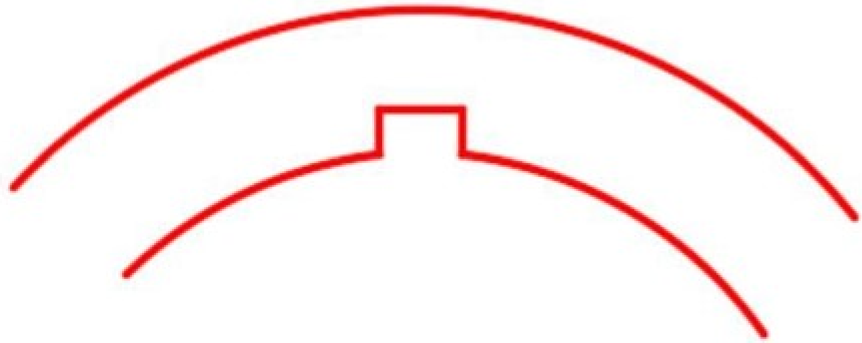


Figure Step 2D
Keyway is Flat Across the Top

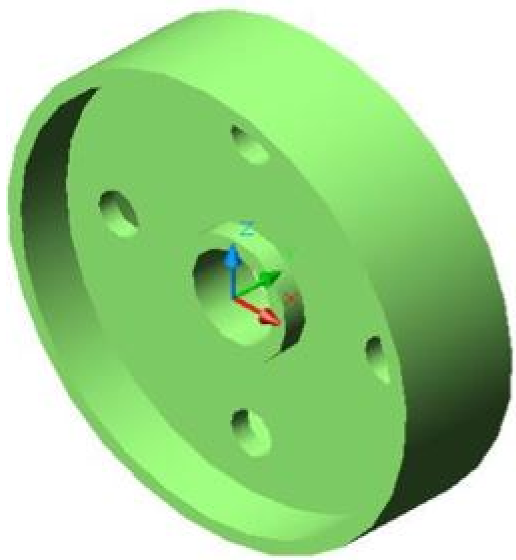


Figure Step 2E
Solid Model – Home View

Step 3

Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches.

Step 4

Apply the colour shown above. (Figure Step 4A and 4B)

Figure Step 4A Completed Model – Home View

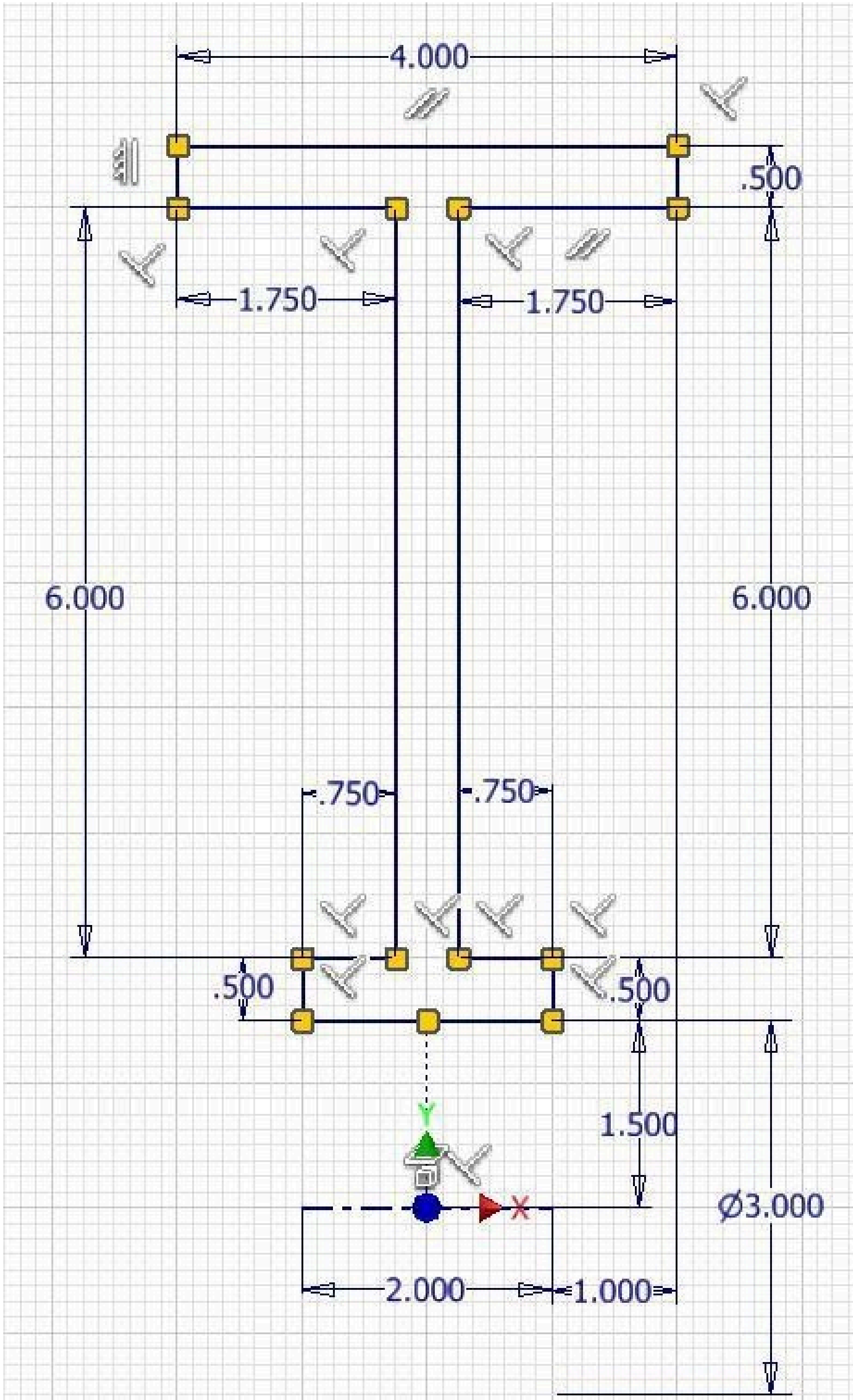
Figure Step 4B Completed Model – Rotated View

Step 5

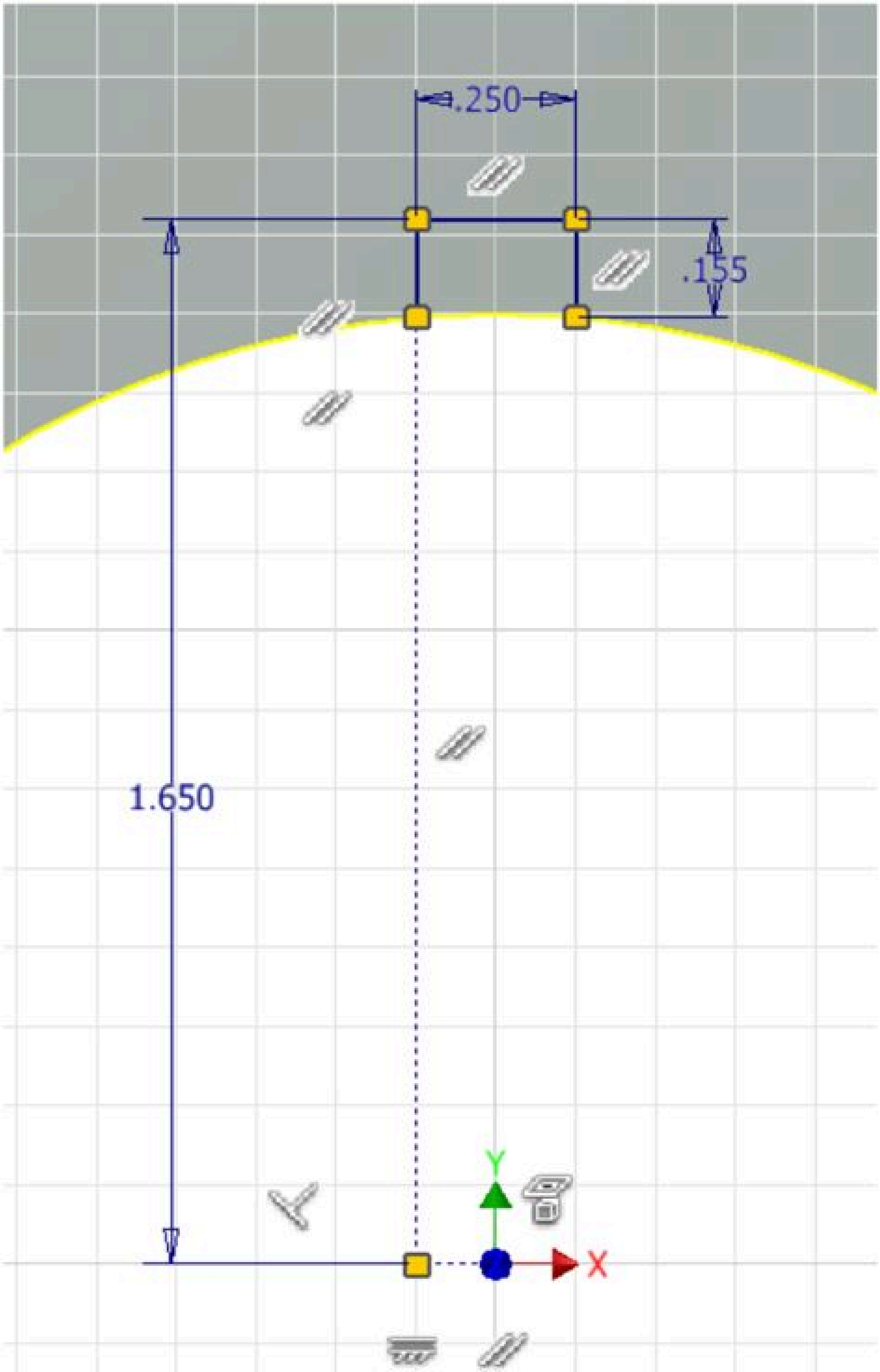
Add the four small holes and the key on new sketches and extrude them.

AUTHOR'S GEOMETRIC CONSTRAINS: The following figures shows the sketch's construction method plus geometric and dimensional constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketch using a different construction method and constraints, that is what is important. You may want to compare your construction method and constraints with the authors.

AUTHOR'S COMMENTS: Your geometrical and dimensional constrains may not match the figure exactly. Just ensure that your sketch is fully constrained.


4.000
.500
1.750
1.750
6.000
6.000
.750
.750
.500
.500
1.500
Ø3.000
Y
X
2.000
1.000

Base Sketch



Key Sketch

Module 15 Fillet and Chamfer Features

Learning Outcomes

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

1. 1 Describe fillet and chamfer features.
2. Apply the FILLET and CHAMFER commands to create fillets and chamfers on solid models.

Fillets

A *fillet* is a tangent arc. A fillet is simply an arc which is tangent at both ends. It can be tangent to two lines, a line and an arc, or two arcs. Technically, it is a fillet when material is added to the object model and a *round* is when material is removed from the object. See Figure 15-1 and 15-2. Most CAD systems, including Inventor, use the term fillet for both.

There are two basic methods of inserting fillets in Inventor. In this module, inserting the fillets after the solid model is created will be taught. They are called features. In the Inventor Advanced book, drawing fillets on the 2D sketch will be taught. It is always better to insert the fillets as features since that makes them much easier to edit after the solid model is created.

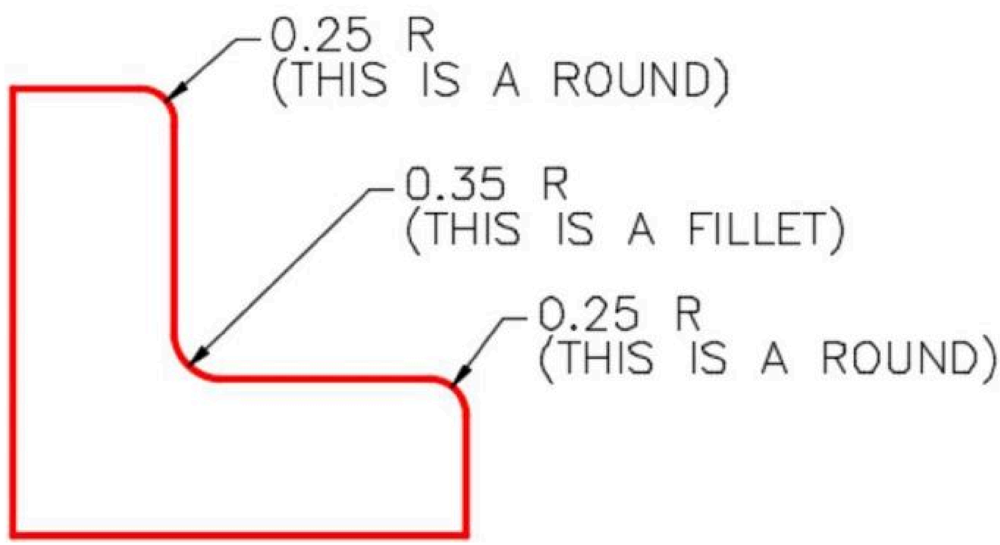


Figure 15-1
Fillets Dimensioned in 2D

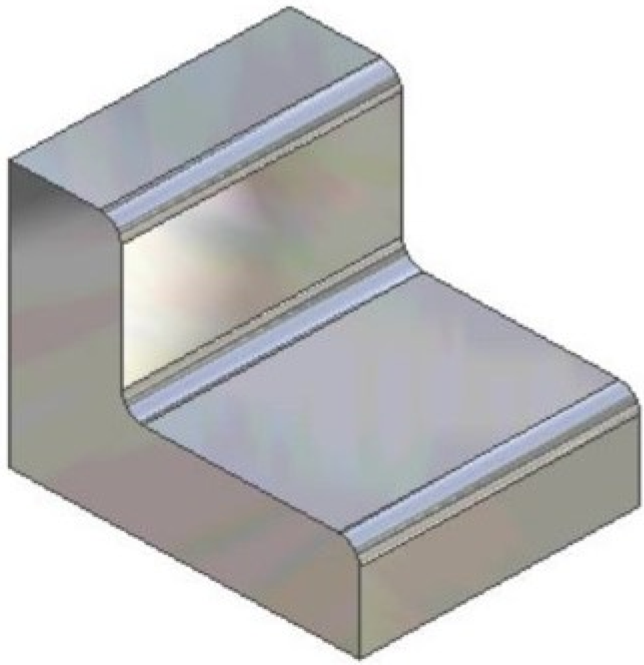


Figure 15-2
How Fillets Appear on the Solid Model

Geometry Lesson Tangency – Part 1

A *point of tangency* is the theoretical point where a line joins an arc or where two arcs join each other making a smooth transition. A line tangent to a circle passes the circle and touches it on only one point on the circle. The point where they touch is called the point of tangency. See Figure 15-3.

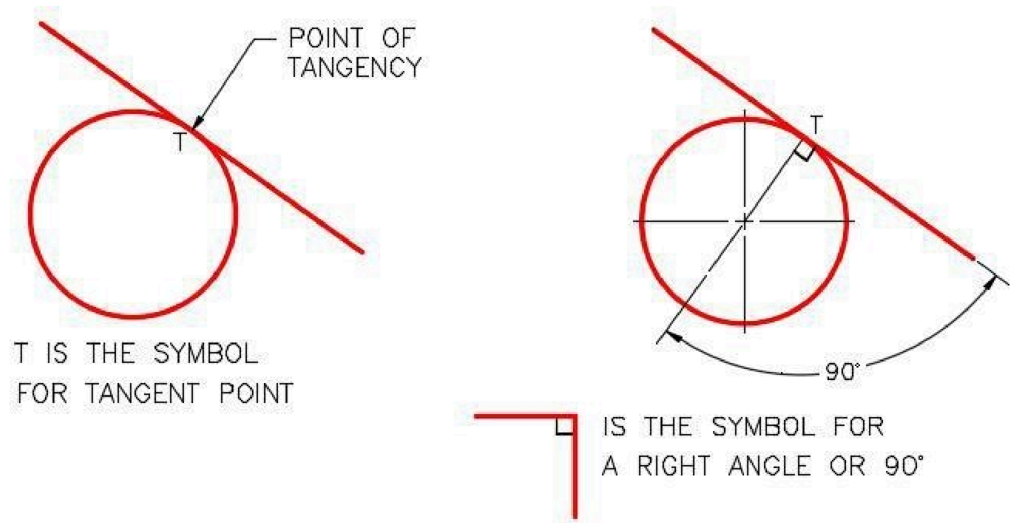
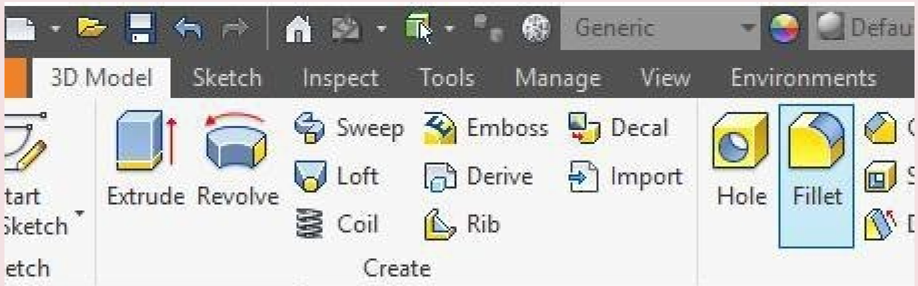


Figure 15-3
Tangency – Part 1

Inventor Command: FILLET

The FILLET command is used to insert a fillet feature on a solid model.

Shortcut: **F**



WORK ALONG: Creating Models With Fillet Features

Step 1

Check the default project and if necessary, set it to Inventor Course.

Step 2

Enter the NEW command to start a new part file using the template: English-Modules Part (in).ipt.

Step 3

Save the file with the name: Inventor Workalong 15-1. (Figure Step 3A and 3B)

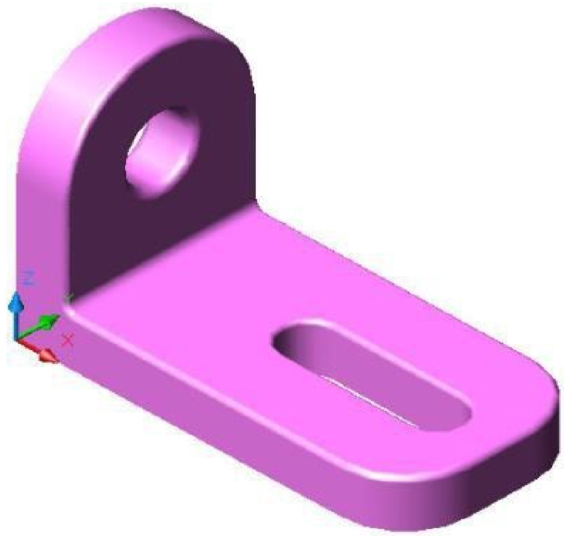


Figure Step 3A
3D Model

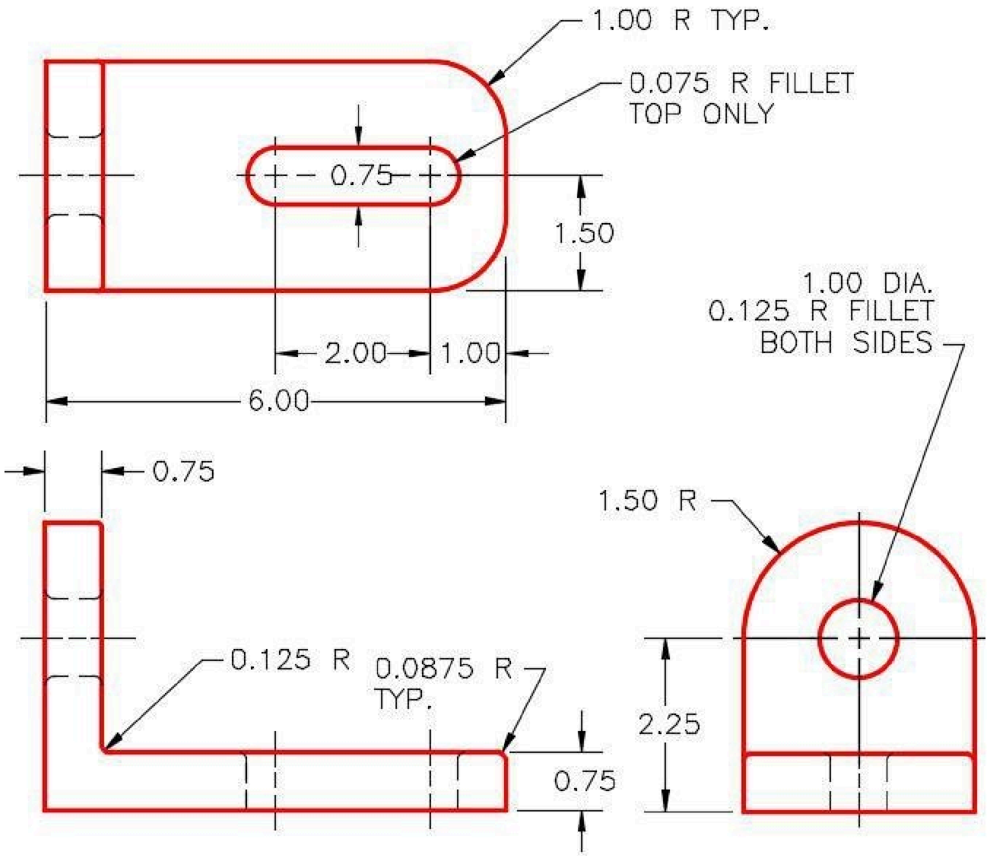


Figure Step 3B

Step 4

Draw the Base sketch on the Front view. Ensure that it is fully constrained. Extrude the sketch to create the Base model. (Figure Step 4A and 4B)

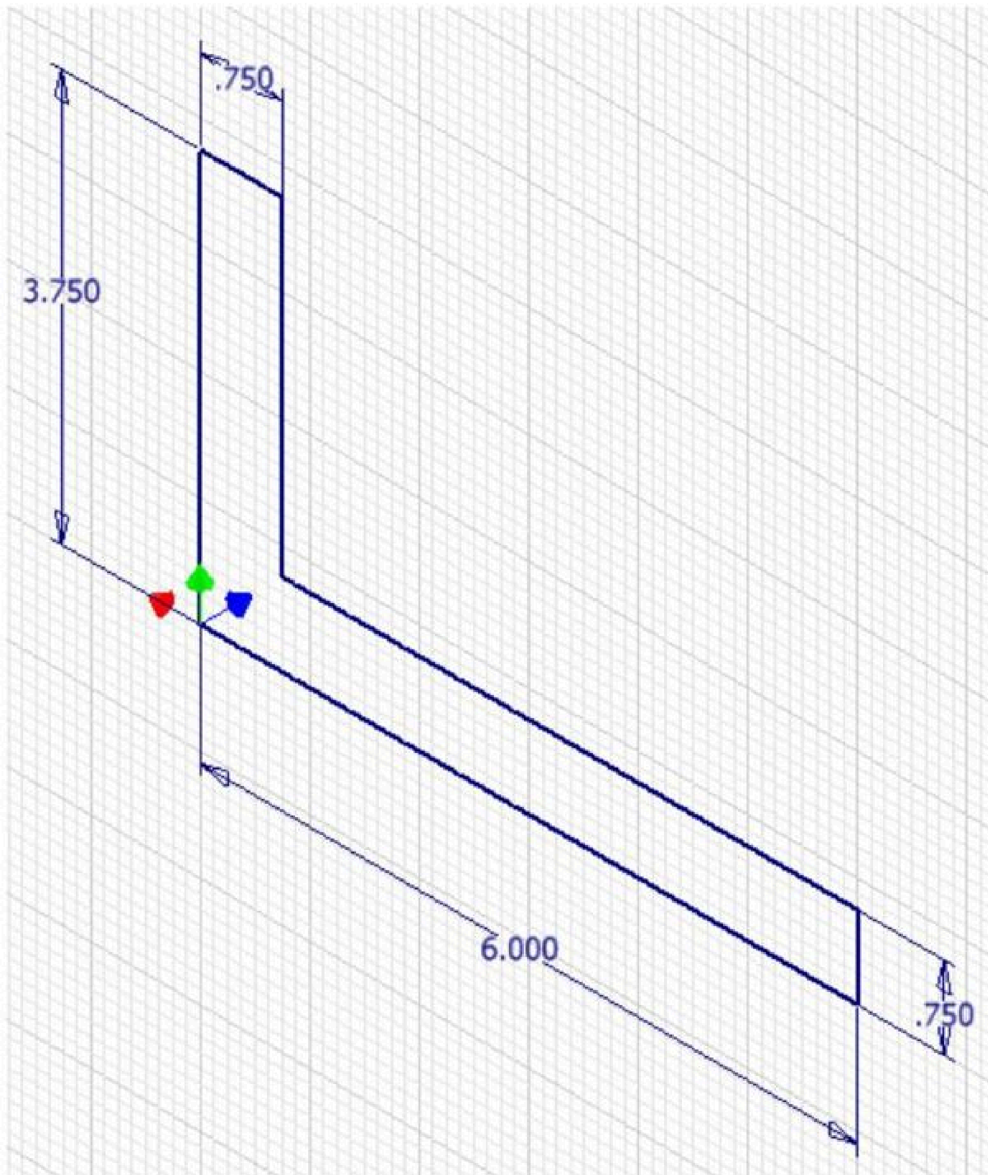


Figure Step 4A

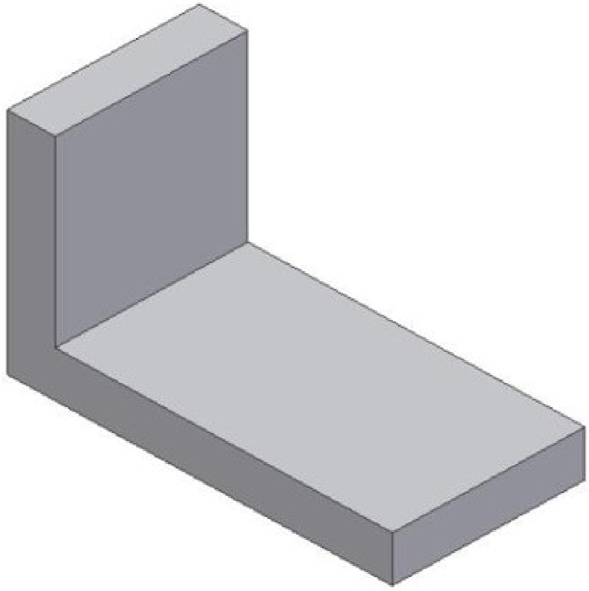


Figure Step 4B

Step 5

Enter the FILLET command and in the Fillet dialogue box, click the radius to edit it. Change the radius to 1.50. (Figure Step 5)

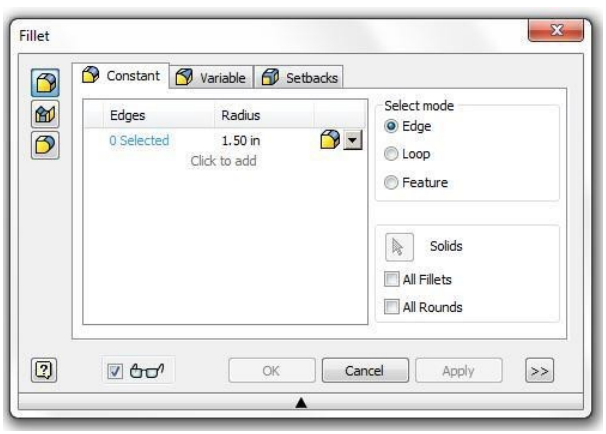


Figure Step 5 [Click to see image full size]

Step 6

Click the Pencil icon to change to the Arrow icon. When the Plus icon appears beside the cursor, select the top left corner of the model. (Figure Step 6A and 6B)

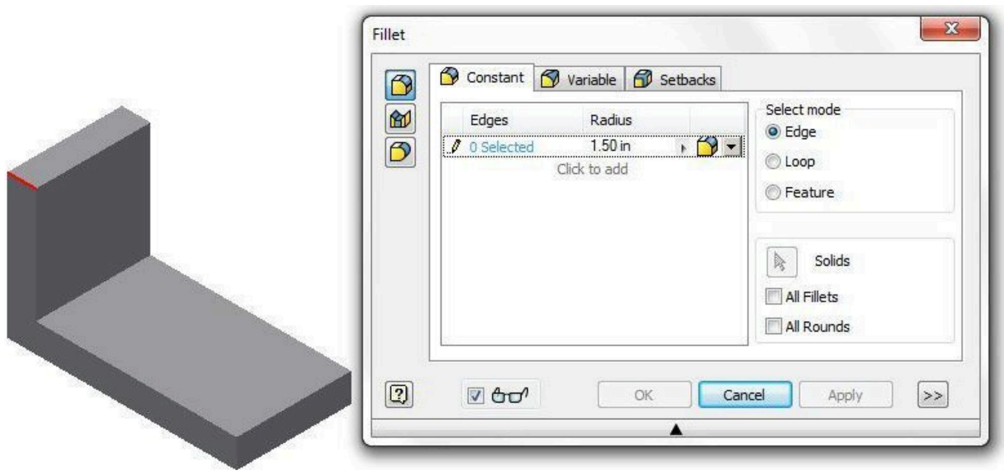


Figure Step 6A [Click to see image full size]

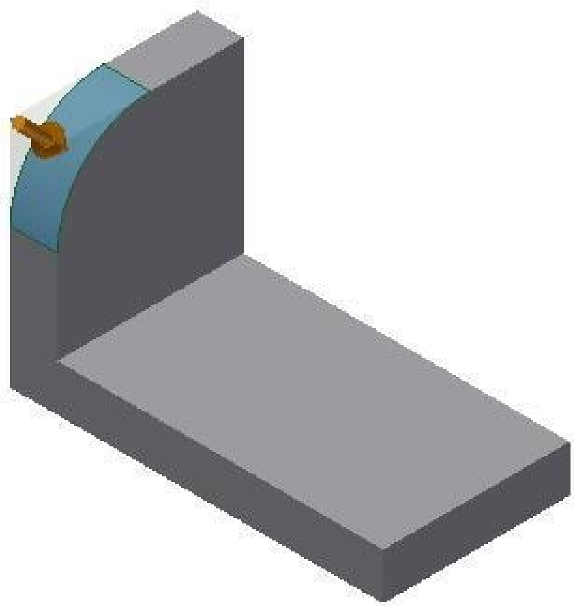
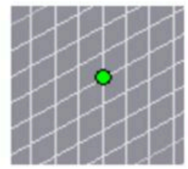


Figure Step 6B

AUTHOR'S COMMENTS: This is the Top view or the XY plane since that is Inventor's default plane. Note that Sketch1 is always on the default plane.

USER TIP: Another reason that it is best to create fillets on the solid model rather than on the skech is that you can then insert them as the last feature to complete the model. The reason that this is important is once the fillet is created the corners are lost on the model. Without the corners, there is nothing to measure from when placing geometry.

Snap Symbols		
Mode	**Icon**	**How they appear in Inventor**
Center (Snaps to the center point of an arc or circle)	N/A	

Step 7

Insert the same fillet on the opposite side of the model. (Figure Step 7)

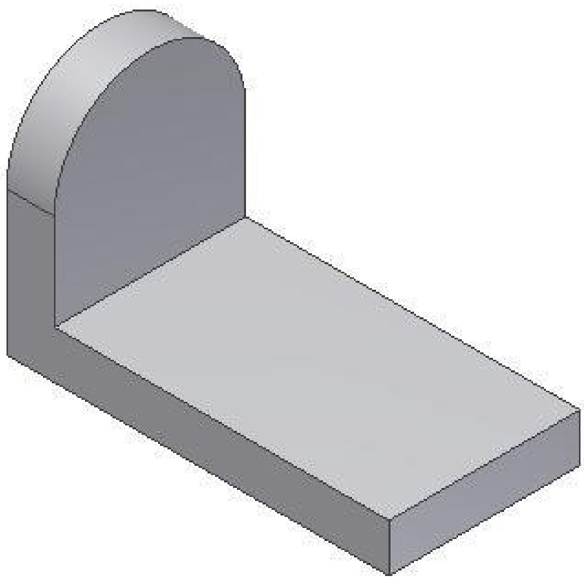


Figure Step 7

Step 8

Start a new sketch and enter the CENTER POINT CIRCLE command. Right click the mouse. In the Right-click menu, select Center. Select the arc and draw a 1 inch Diam circle. Dimension and extrude it.(Figure Step 8A, 8B, 8C, and 8D)

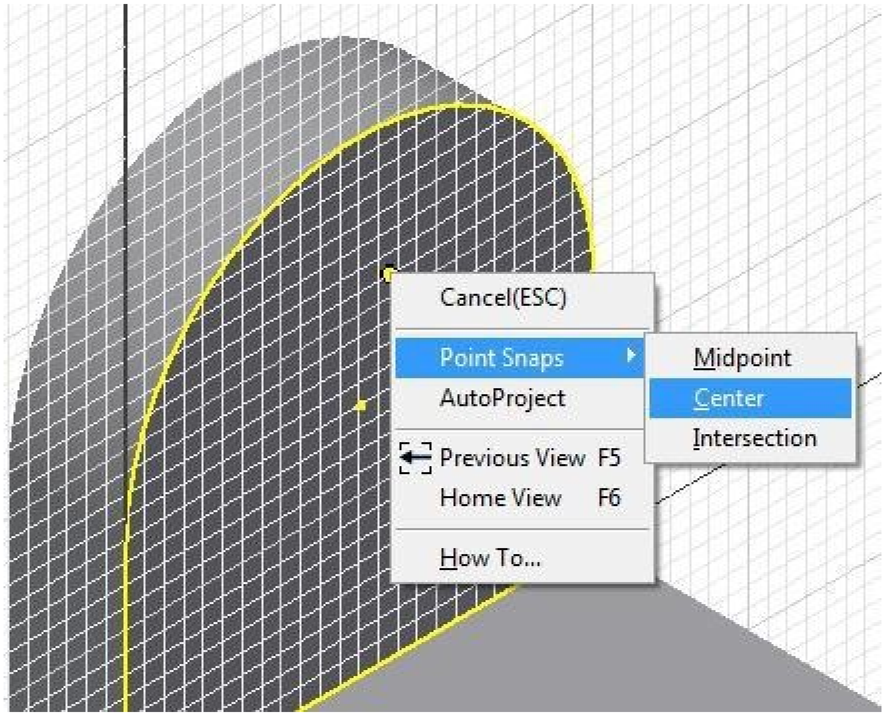


Figure Step 8A

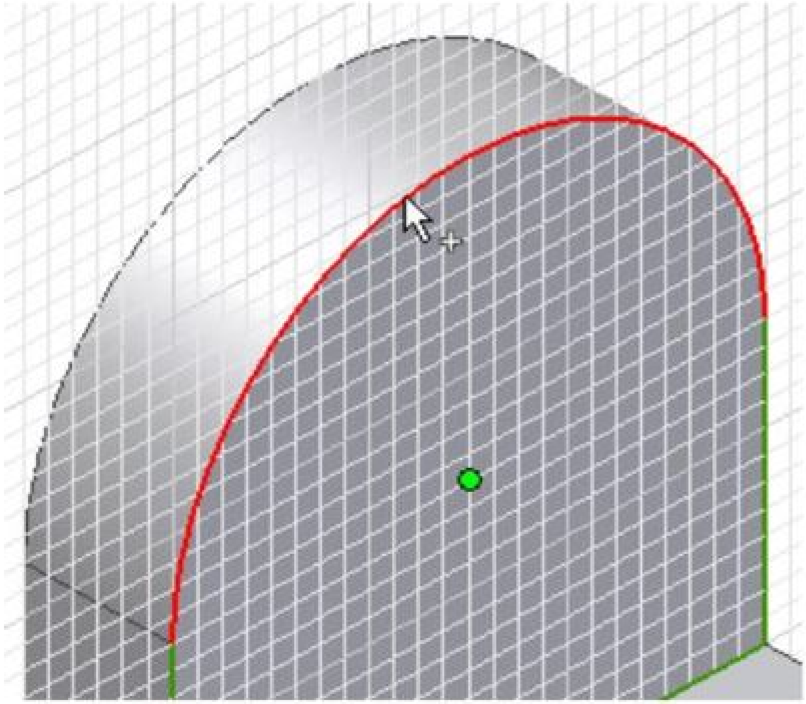


Figure Step 8B

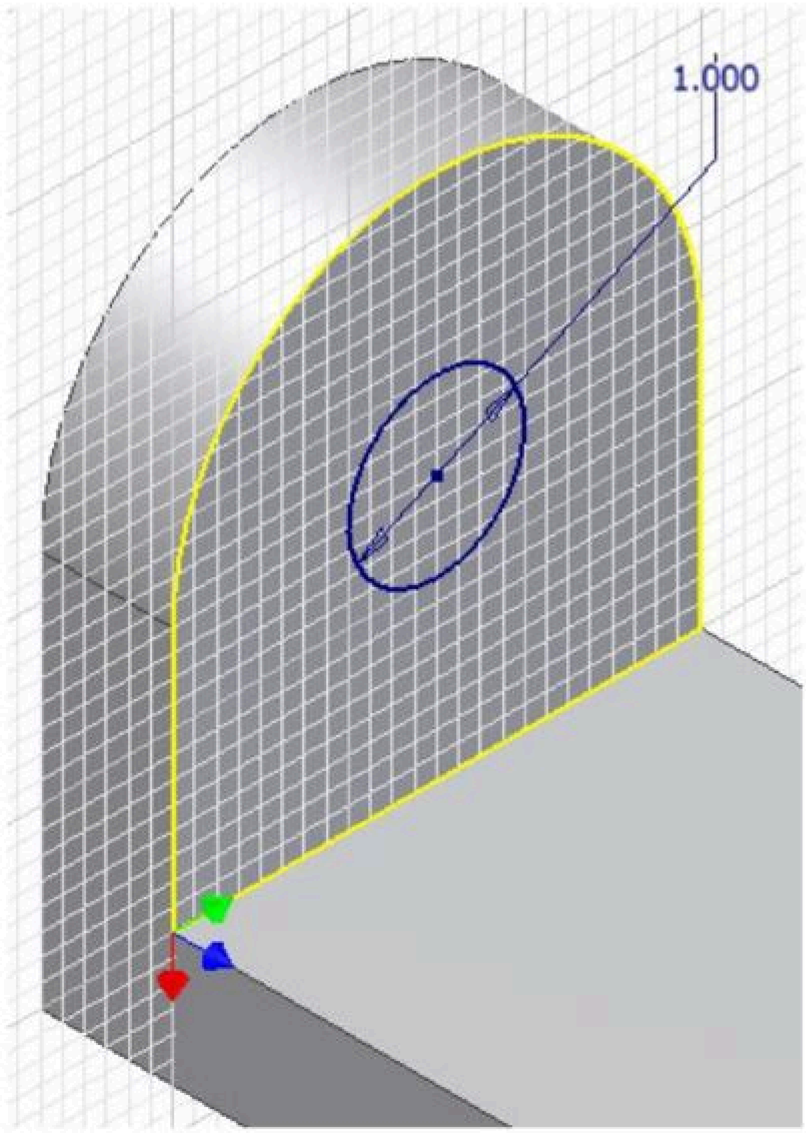


Figure Step 8C

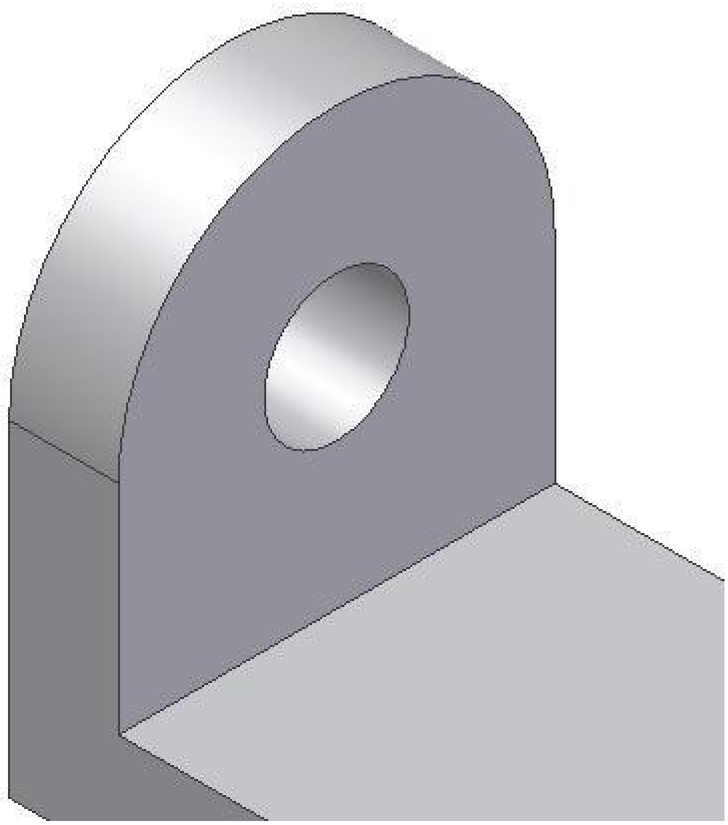


Figure Step 8D

MUST KNOW: A fillet is a tangent arc. A fillet is simply an arc that is tangent at both ends. It can be tangent to two lines, a line and an arc, or two arcs but it must be tangent at both ends. Technically, a fillet is when material is added to the object. It is called a round when material is removed from the object.

Step 9

Start a new sketch. On it, draw and dimension the slot and extrude it. (Figure Step 9A and 9B)

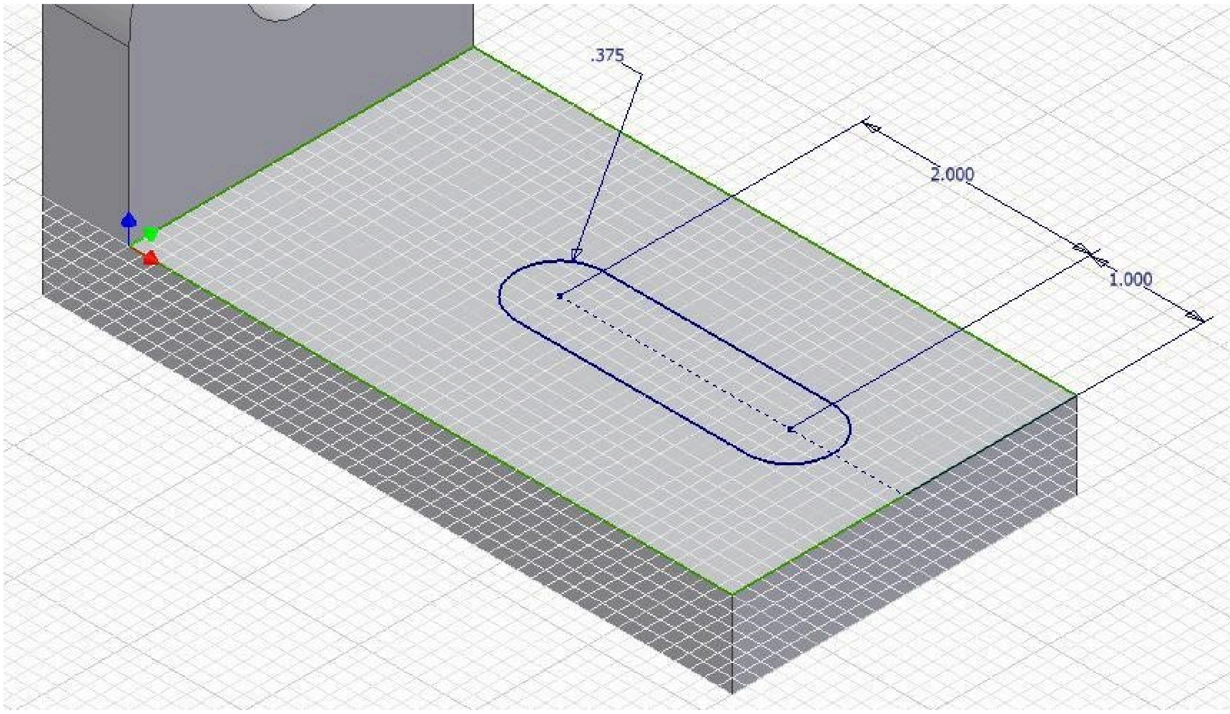


Figure Step 9A [Click to see image full size]

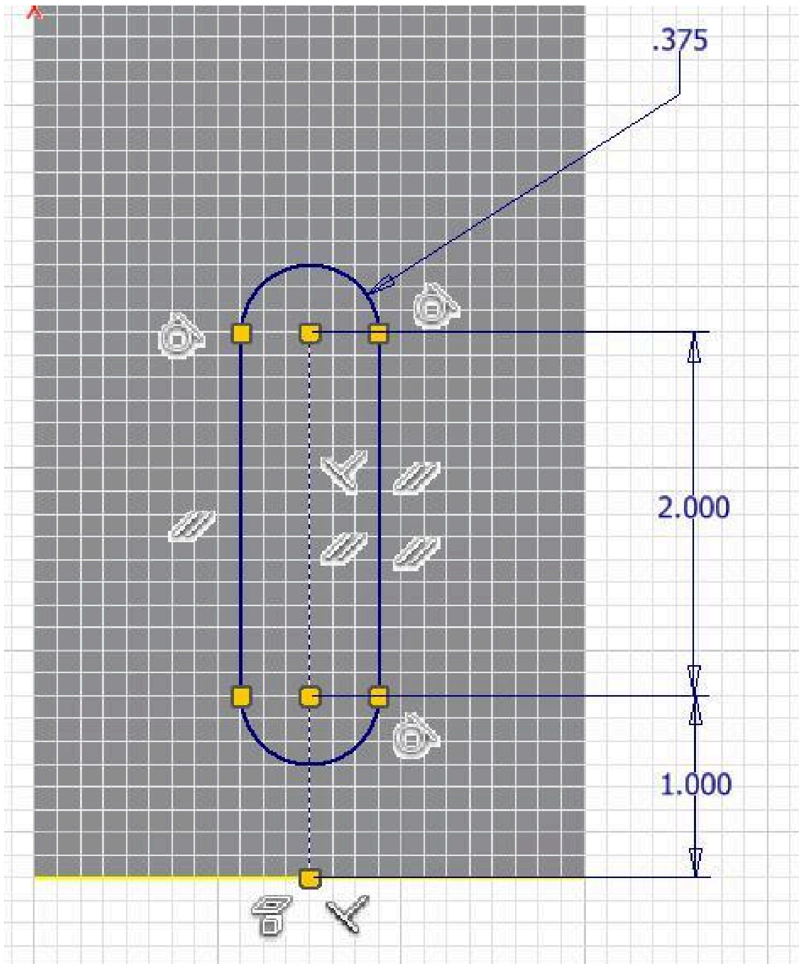


Figure Step 9B

Step 10

Using what you just learned, insert the 1 inch radius fillets. (Figure Step 10)

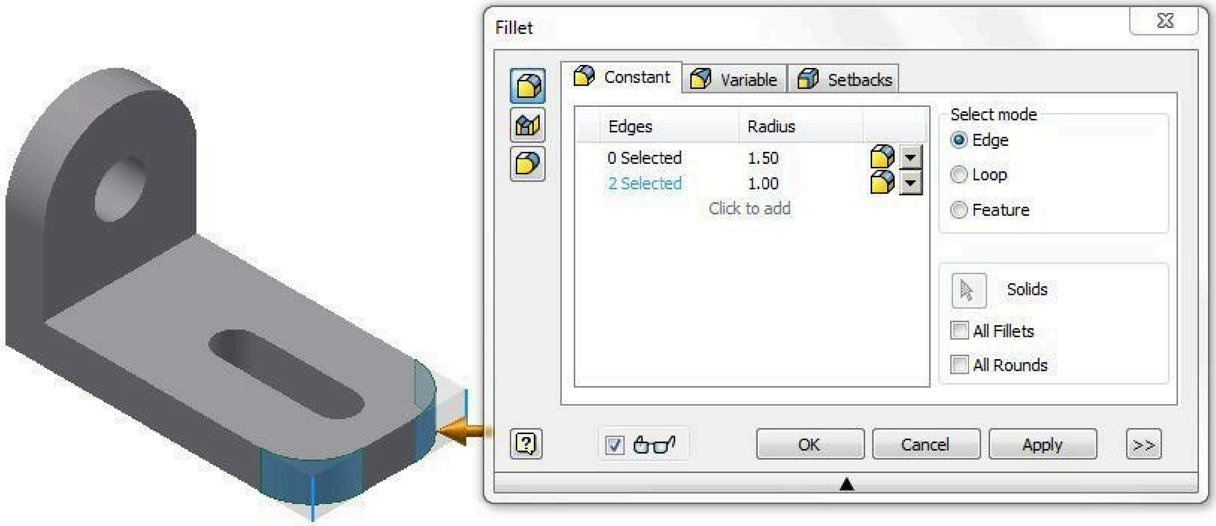


Figure Step 10 [Click to see image full size]

Step 11

Insert the fillets as shown on the figures. Ensure that you set the correct radius for each fillet. (Figure Step 11A and 11B)

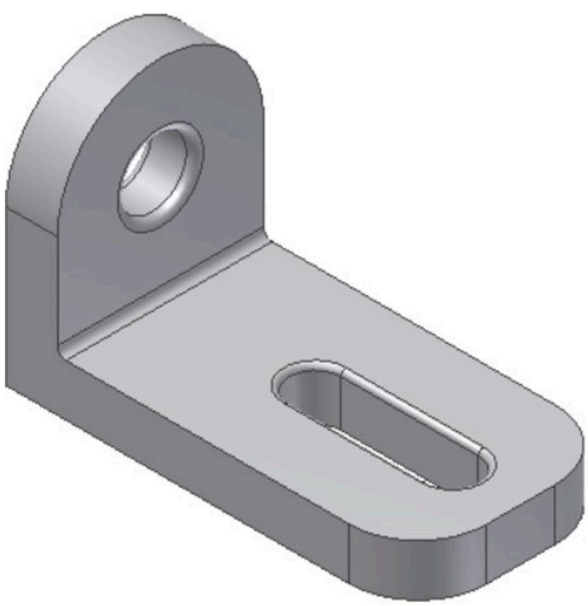


Figure Step 11A

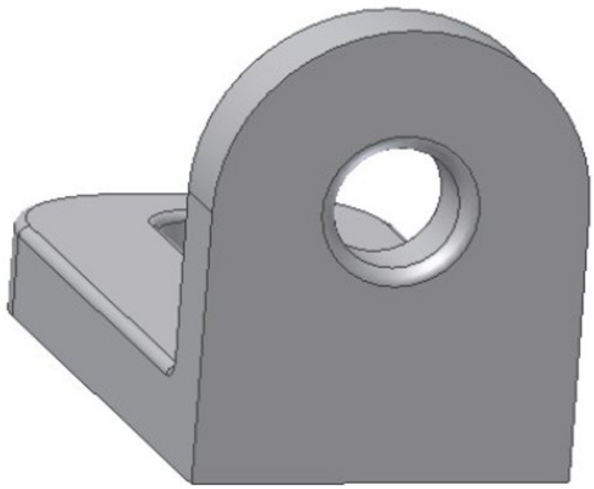


Figure Step 11B

Step 12

Enter the FILLET command and in the Fillet dialogue box, set the Radius to 0.0875. Set the Select mode to Loop and select the edge as shown in the figure. (Figure Step 12)

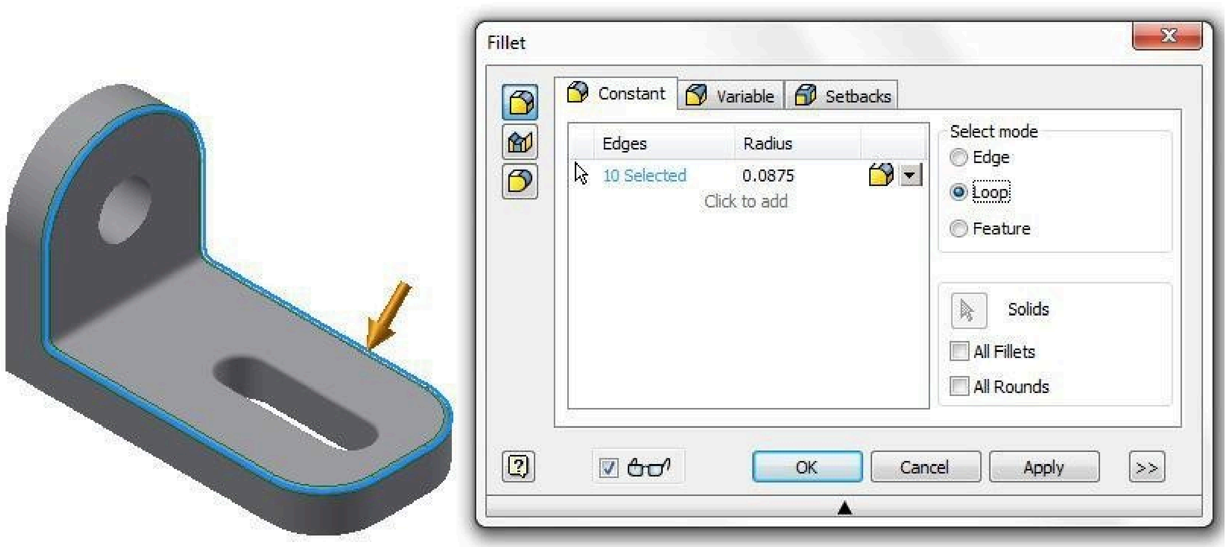


Figure Step 12 [Click to see image full size]

Step 13

To complete the model, change the colour to: Chrome – Polished Black. (Figure Step 13)

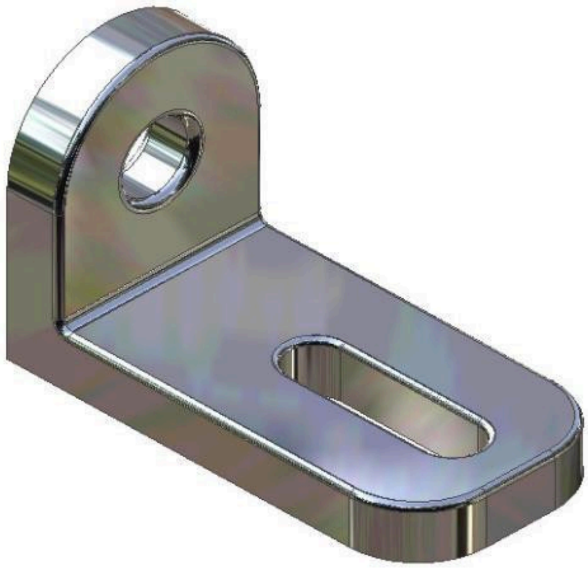


Figure Step 13

Step 14

Save and close the file.

USER TIP: Inventor allows two methods of inserting fillets and chamfers. The first method is to create the solid model using extrude or revolution just as you have being doing to this point in the course and then insert the fillets on the solid model.The second method is drawing the fillets in the 2D sketch and they will be created when the sketch is extruded or revolved. The first method is the BEST method and should be used whenever possible. One reason that it is the best method is it allows you to decide the order of filleting. This is especially important where two or more fillets meet or intersect on the model. This is the method you will be using in this module.

Chamfers

A *chamfer*is similar to a fillet except instead of an arc being inserted, it inserts an inclined line. See Figure 15-4. Figure 15-5 shows a multiview drawing with the chamfers dimensioned. There are two basic methods of inserting chamfers in Inventor, the same as fillets. In this module, you will learn how to insert them as features after the solid model is created as shown in Figure 15-6.

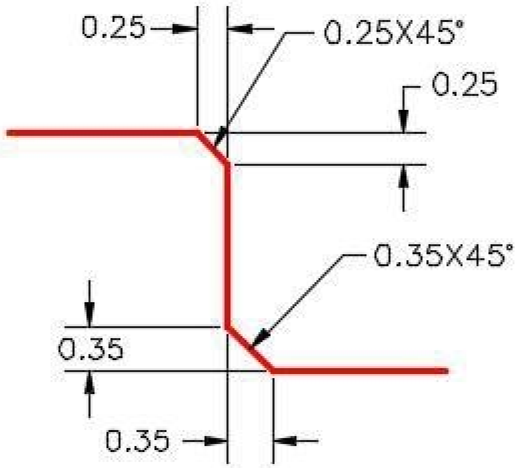


Figure 15-4
Chamfers

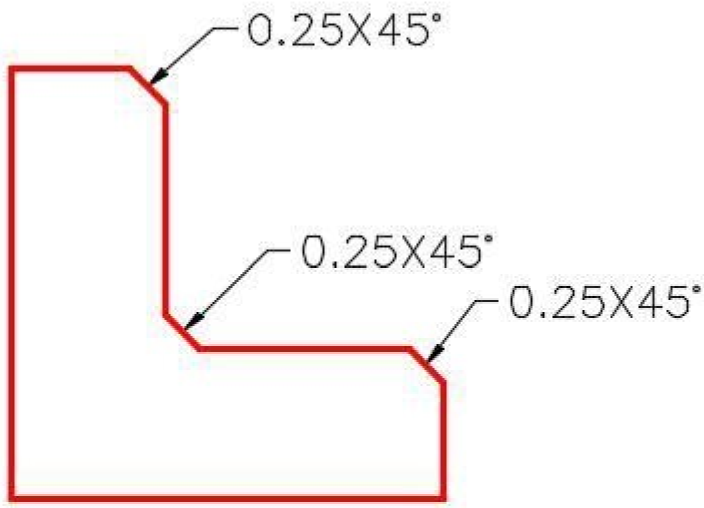


Figure 15-5
Chamfers in a Multiview Drawing

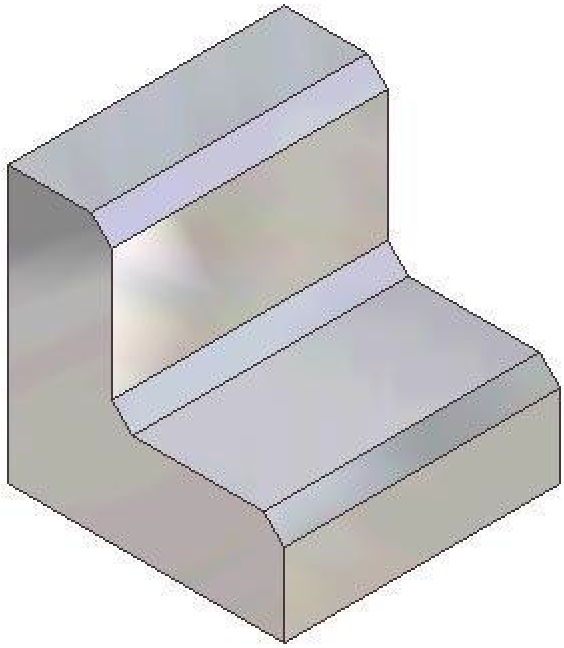
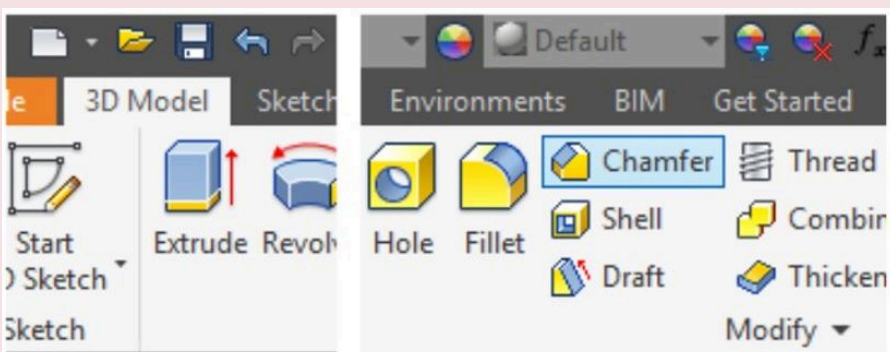


Figure 15-6
Chamfers on the Solid Model

Inventor Command: CHAMFER

The CHAMFER command is used to create a chamfer feature on a solid model.

Shortcut: **CTRL+ Shift+K**



WORK ALONG: Creating Models with Chamfer Features

Step 1

Open the file: <u>Inventor Lab 13-1.ipt</u> that you created in Module 13.

Step 2

Using the SAVEAS command, save the file with the name: Inventor Workalong 15-2. (Figure Step 2A and 2B)

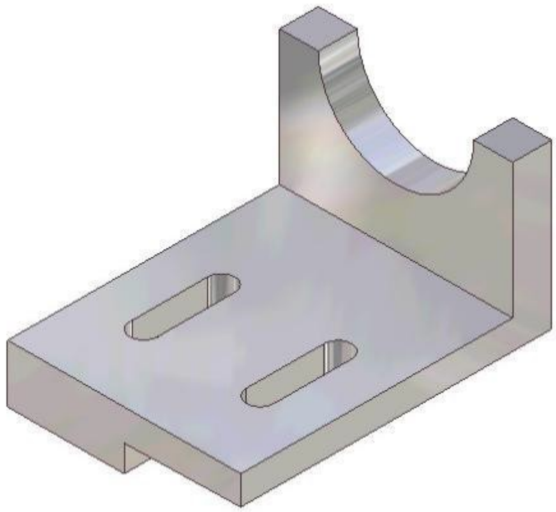


Figure Step 2A

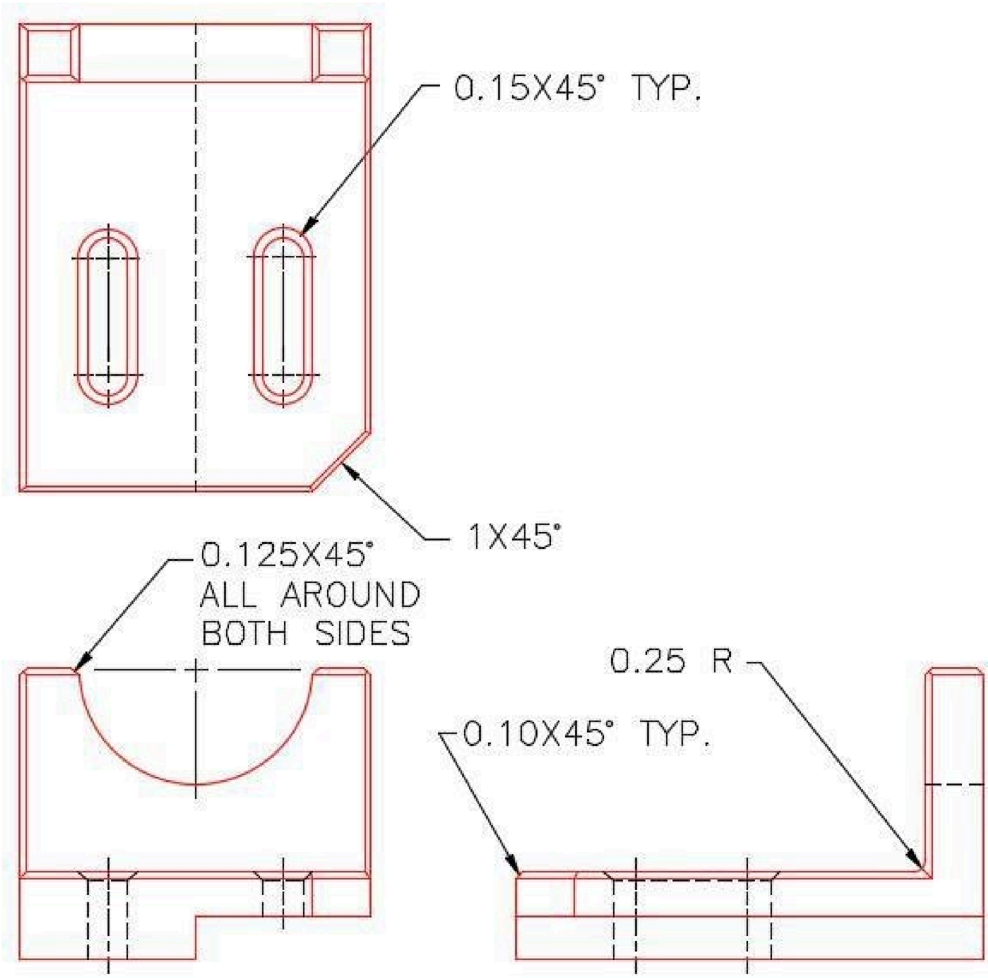


Figure Step 2B
Dimensioned Multiview Drawing

Step 3

Enter the CHAMFER command. In the Chamfer dialogue box, ensure that the equal side icon and Edges icon are enabled. Set the distance to 1.0. Select the bottom right corner of the model. (Figure Step 3)

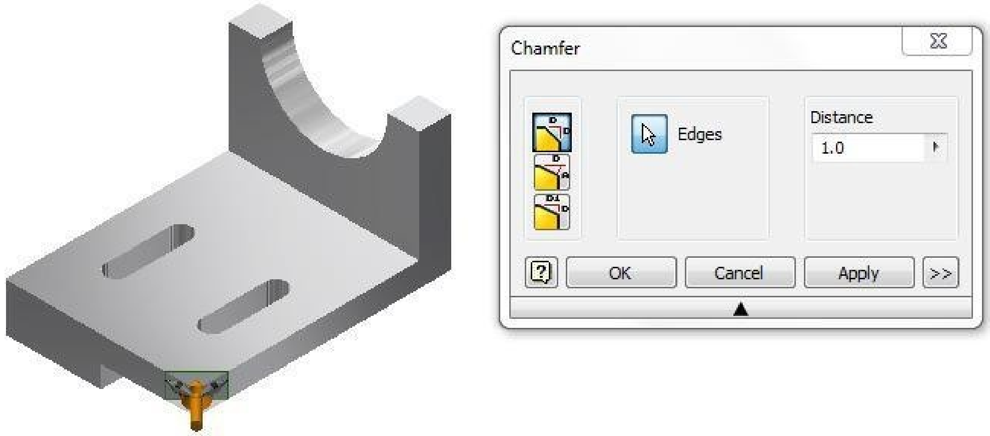


Figure Step 3 [Click to see image full size]

Step 4

Set the distance to 0.125 and insert the chamfer on the around the top as shown in the figure. (Figure Step 4)

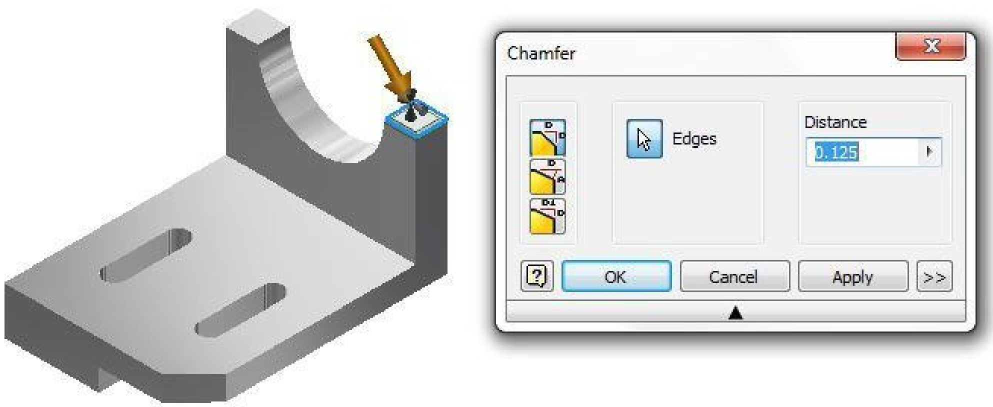


Figure Step 4 [Click to see image full size]

Step 5

Using what you just learned, insert the fillet and chamfers on the model as shown in the figure. (Figure Step 5)

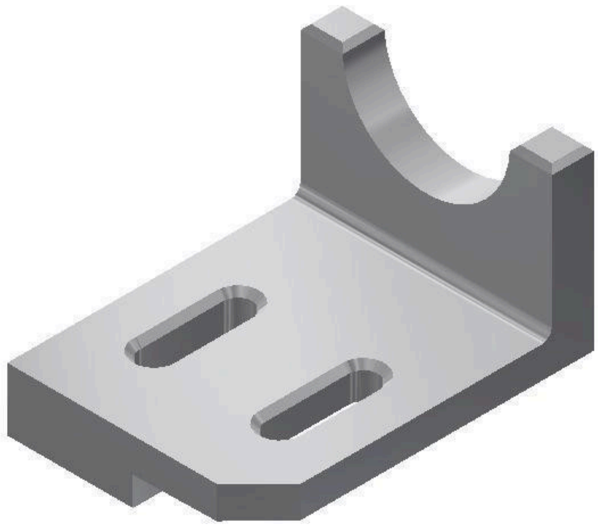


Figure Step 5

Step 6

Insert the 0.10 chamfers around the outside as shown in the figure. (Figure Step 6)

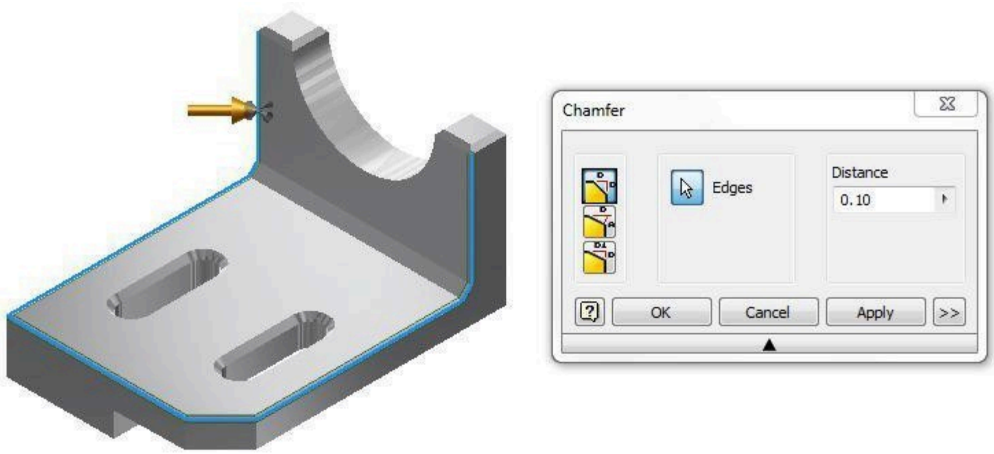


Figure Step 6 [Click to see image full size]

Step 7

The completed model should appear as shown in the figure. (Figure Step 7)

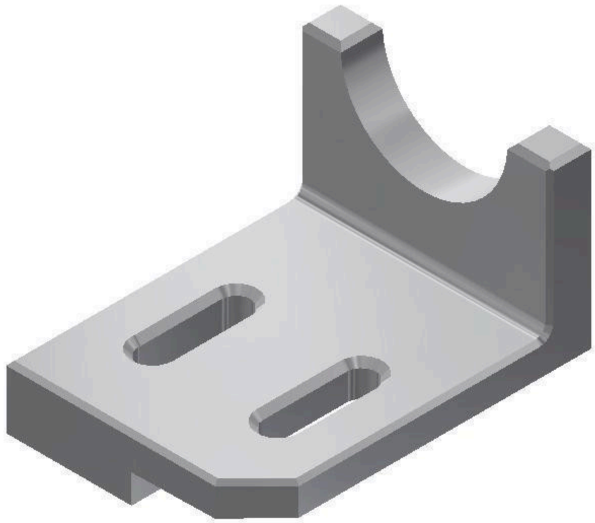


Figure Step 7

Step 8

Save and close the file.

USER TIP: Many commands can be used while you are activity using another command. The ORBIT command is one of them. To increase your drawing speed, you can preform an operation on one side of a model and then, while you are still in the command, press F4 and rotate the model. Then preform the operation on another side. This would have worked well for Step 11 as you could have placed the fillet on the other side of the hole in the same FILLET command.

Key Principles

Key Principles in Module 15

1. A fillet is a tangent arc. A fillet is simply an arc which is tangent at both ends. It can be tangent to two lines, a line and an arc or two arcs.
2. A chamfer is similar to a fillet except instead of an arc being inserted, it creates an inclined line.
3. There are two basic methods of inserting fillets in Inventor. In this module, inserting the fillets after the solid model is created will be taught. They are called features. In the Inventor Advanced Modules, drawing fillets on the 2D sketch will be taught. It is always better to insert the fillets as

features since that makes them much easier to draw and edit after the solid model is created.

Lab Exercise 15-1

Time allowed: 60 minutes.

Part Name	**Project**	**Units**	**Template**	**Color**	**Material**
Inventor Lab Lab 15-1	Inventor Course	mm	Metric-Modules Part (mm).ipt	Steel – Galvanized	N/A

Step 1

Project the Center Point onto the Base plane.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude them to produced the solid model shown in the figures. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. (Figure Step 2A, 2B, and 2C)

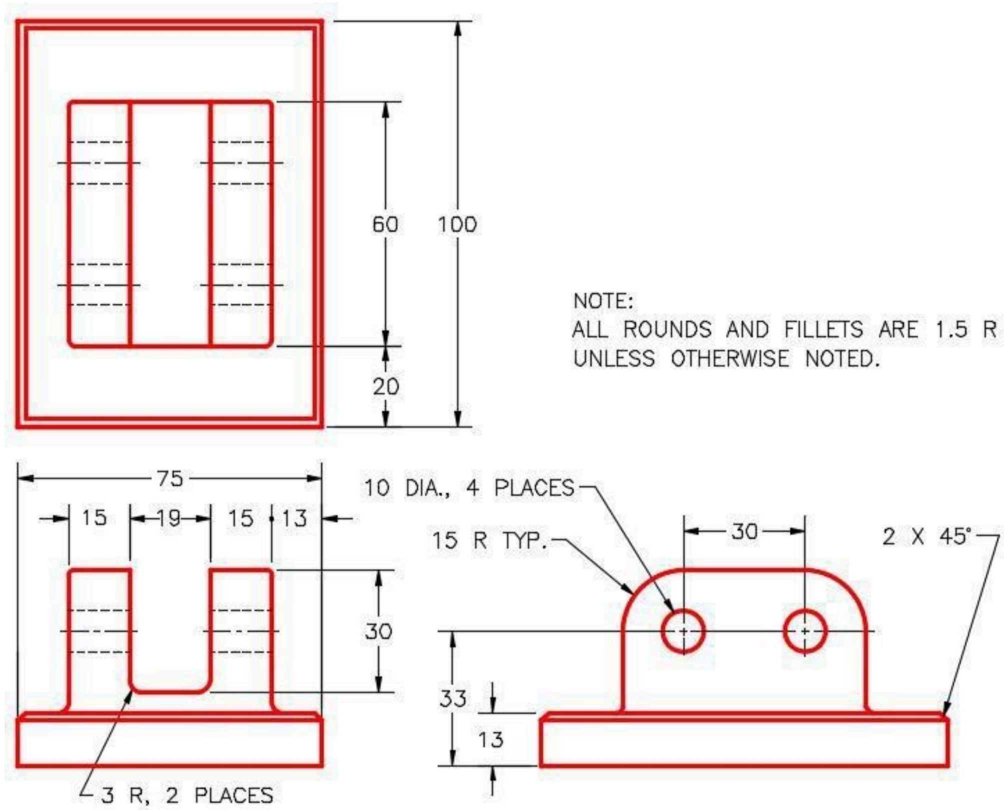


Figure Step 2A
Dimensioned Multiview Drawing

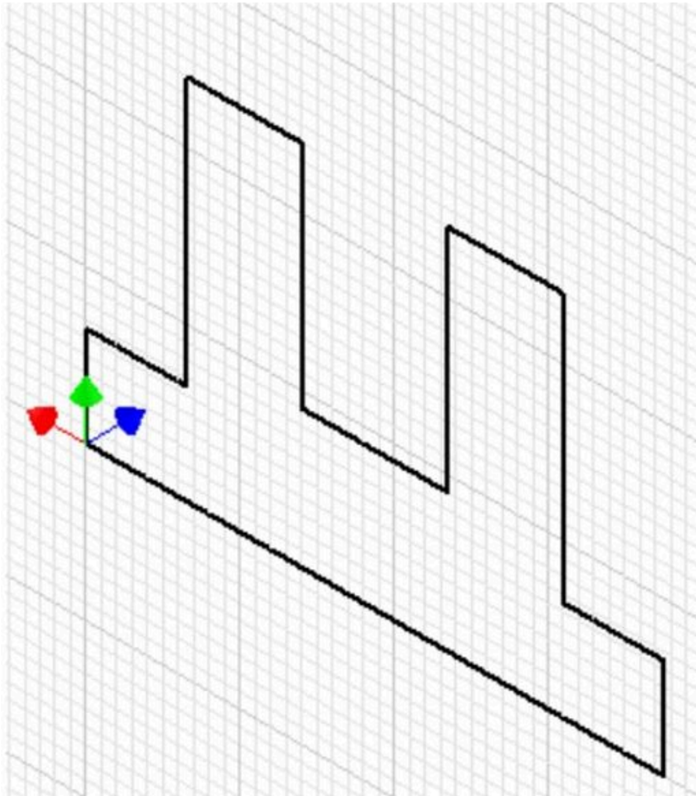


Figure Step 2B
Suggested Base Sketch –
Front – XZ Plane

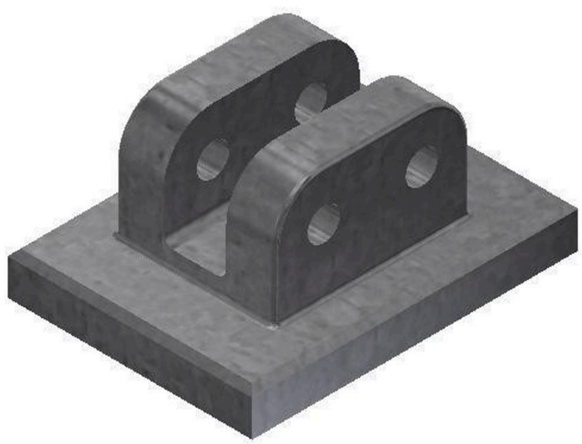


Figure Step 2C
3D Model – Home View

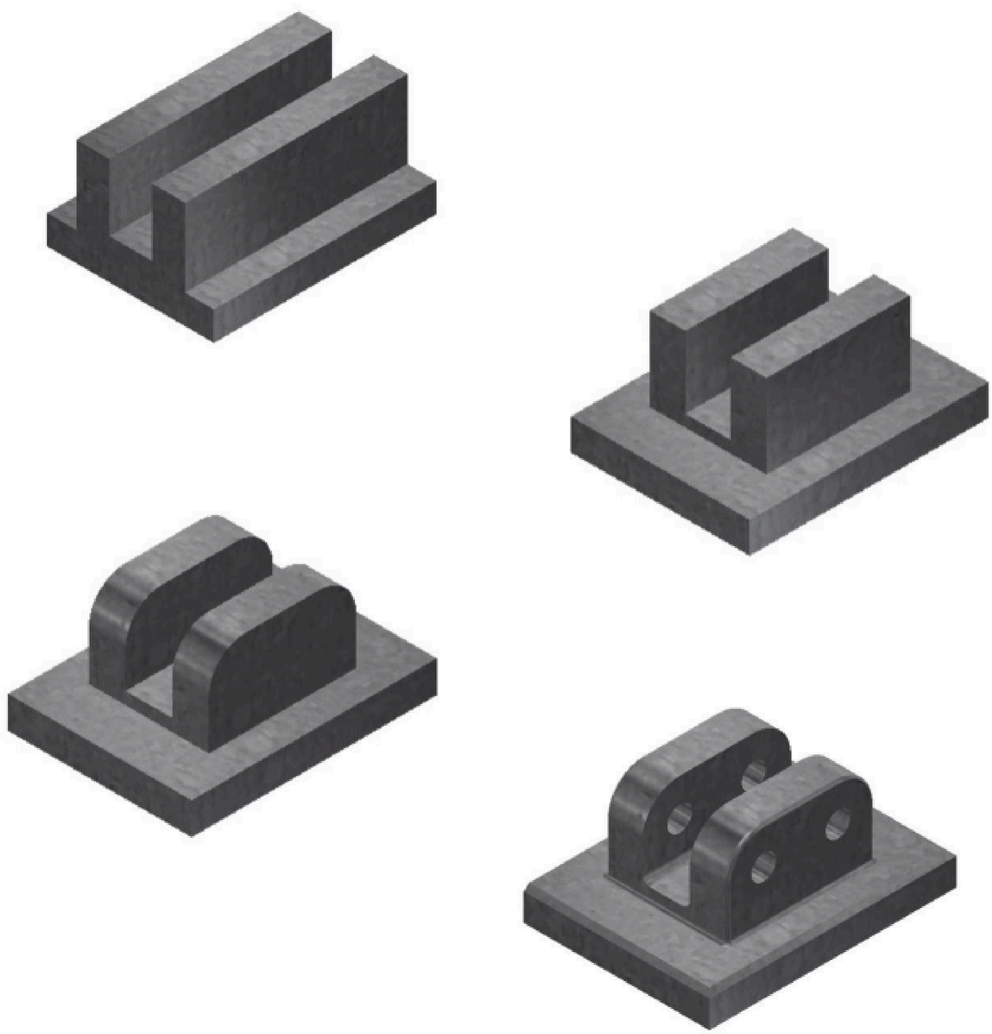
Step 3

Insert the fillets and chamfers as features after the model is constructed.

Step 4

Apply the colour shown above.

AUTHOR'S GEOMETRIC CONSTRAINS: The following figures shows the construction method suggested by the author to help you learn how to construct models. It is only the suggested method and if you can complete a fully constrained sketch and complete the model, that is what is important. You may want to compare your construction methods with the authors.



Lab Exercise 15-2

Time allowed: 30 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab Lab 15-2	Inventor Course	Inches	N/A	Aluminum – Polished	N/A

Step 1

Open the file: Inventor Lab 14-2.ipt that you created in Module 14.

Step 2

Using the SAVEAS command, save it with the name: Inventor Lab 15-2.

Step 3

Add the fillets and chamfers as features as shown in the figures. (Figure Step 3A, 3B, and 3C)

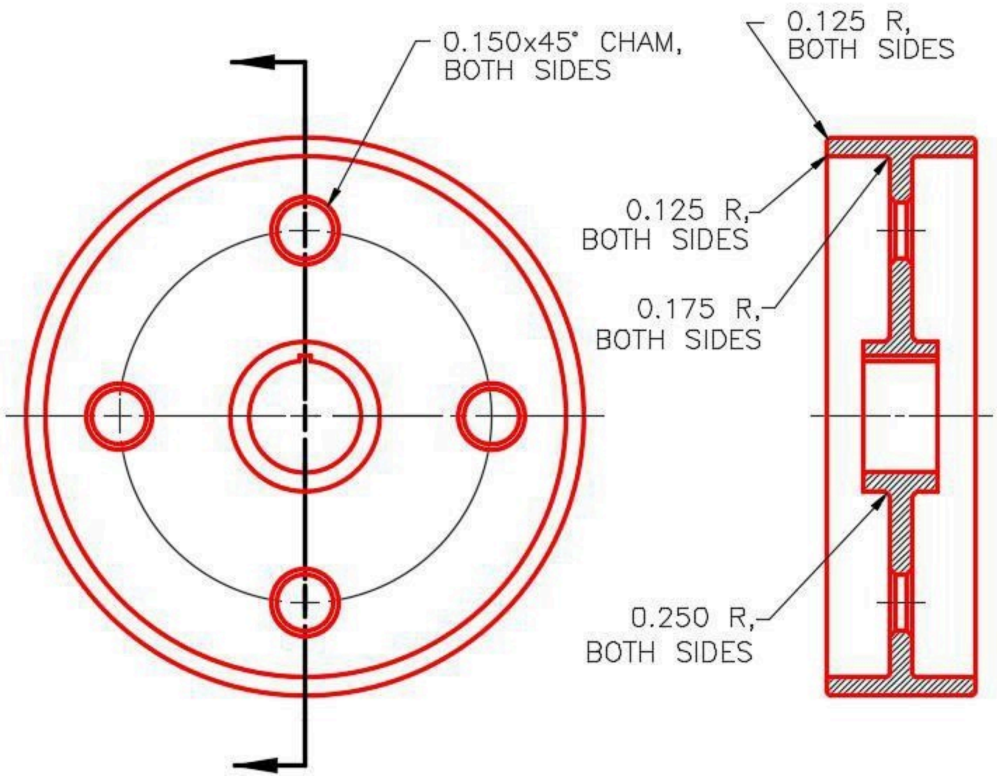


Figure Step 3A
Dimensioned Multiview Drawing

Figure Step 3B
Solid Model –
Home View

Figure Step 3C
Solid Model –
Rotated View

Module 16 Competency Test No.3 Open Book

Learning Outcomes

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

1. Within a two hour time limit, complete a written exam and a lab exercises.

Competency Tests

The Inventor 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 you have completed this module.

Multiple Choice Questions

Select the BEST answer.

1. Which on of the following statements is true?
    A. Construction objects are drawn with the command CONSTRUCTION.
    B. Construction objects will not be used when the sketch is converted into a 3D feature.
    C. Construction objects cannot be used to constrain the sketch.
    D. Construction objects cannot be used as dimensional constraints.
    E. Construction objects will be used when the sketch is converted into a 3D feature.
2. When is it best to place fillets and chamfers on a solid model?

A. The last thing you do to complete the solid.
B. The first thing after the base sketch it is revolved.
C. Before the base model is created.
D. When the sketches are being created.
E. The first thing after the base sketch is extruded.

3. Which snap mode is used to snap to the location where two object cross?
   A. On
   B. Midpoint
   C. Intersection
   D. Point
   E. Center
4. Which one of the following function keys, when pressed, enables the display of the constraint icons?
   A. F2
   B. F4
   C. F6
   D. F8
   E. F10
5. What two commands are used to create a solid model?
   A. REVOLVE AND OFFSET
   B. EXTRUDE and REVOLVE
   C. EXTRUDE and FILLET
   D. PROJECT and SOLID
   E. EXTRUDE and MODEL
6. Which key or keys, when pressed while you are selecting objects, will allow you to select more then one object in the selection set?
   A. Only TAB
   B. Either ALT or SHIFT
   C. Either CTRL or ALT
   D. Either TAB or CTRL
   E. Either CTRL or SHIFT.
7. What does the REVOLVE command do when it finds a centerline in the base sketch it is revolving?
   A. It automatically uses the centerline as the axis of revolution.

B. It prompts you to select the axis of revolution.

C. It extrudes the base sketch using it as the centre.

D. It ignores the centerline.

E. It allows you to select another axis as the axis of revolution.

8. Which snap mode is used when you want to snap to the centre of a line?

    A. On

    B. Midpoint

    C. Intersection

    D. Point

    E. Center

9. Which geometrical constraint is used to make 4 circles all the same diameter when only one of them is dimensioned?

    A. Tangent

    B. Concentric

    C. Coincident

    D. Center

    E. Equal

10. What one of the following is used to change the properties of a selected object from a drawing object to a construction object?

    A. Construction menu

    B. Construction icon

    C. Construction line

    D. Construction circle

    E. Construction object

Lab Exercise 16-1

Time allowed: 60 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab Lab 16-1	Inventor Course	Inches	English-Modules Part (in).ipt	Chrome – Polished Blue	N/A

Step 1

Note the location of X0Y0Z0. Draw the necessary sketches and revolve or extrude them to produce the 3D model. (Figure Step 1)

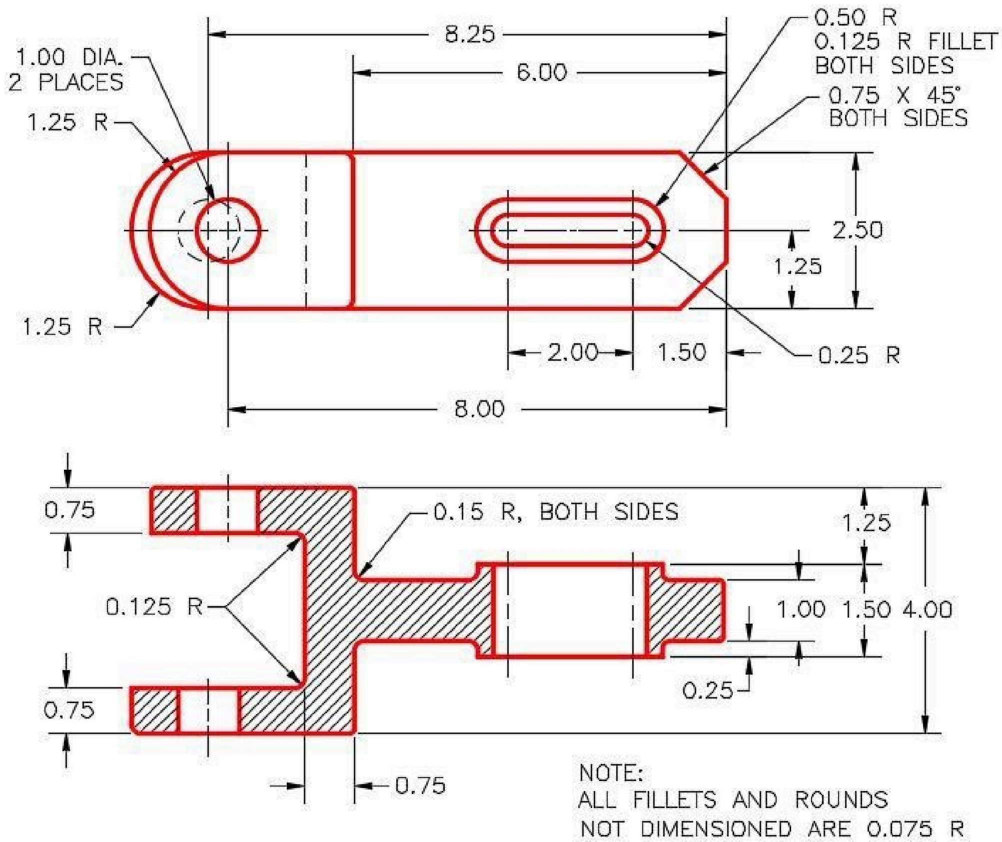


Figure Step 1
Dimensioned Multiview Drawing [Click to see image full size]

Step 2

Draw the base sketch on the Front view. (Figure Step 2)

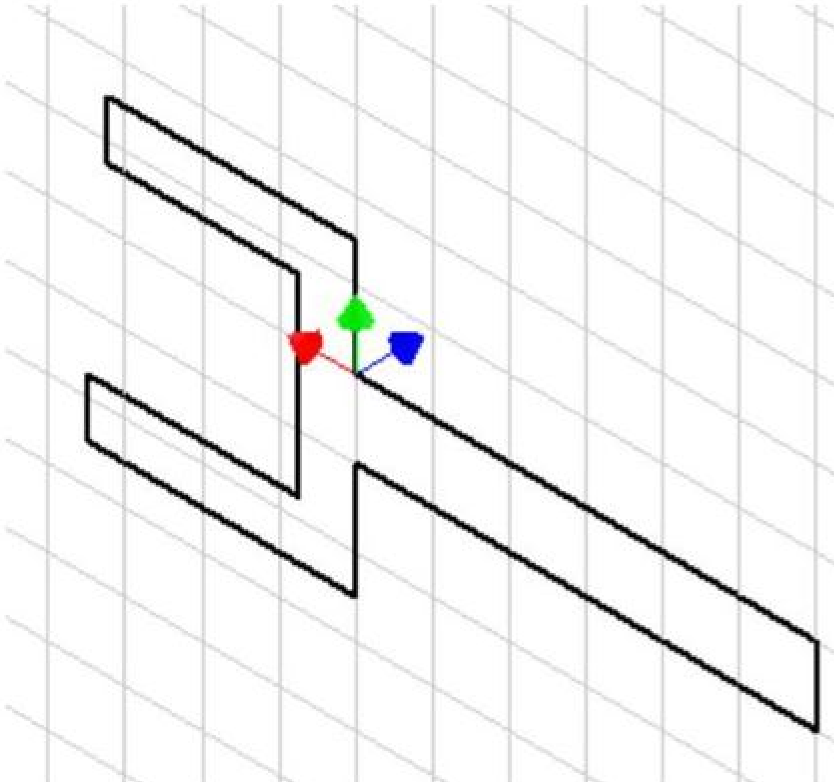


Figure Step 2
Suggested Base Sketch –
Front View (XZ Plane)

Step 3

Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. Ensure that all objects, on all sketches, display purple on a black background.

Step 4

Create the fillets and chamfers after the solid model is totally constructed.

Step 5

Apply the colour shown. (Figure Step 5A, 5B, and 5C)

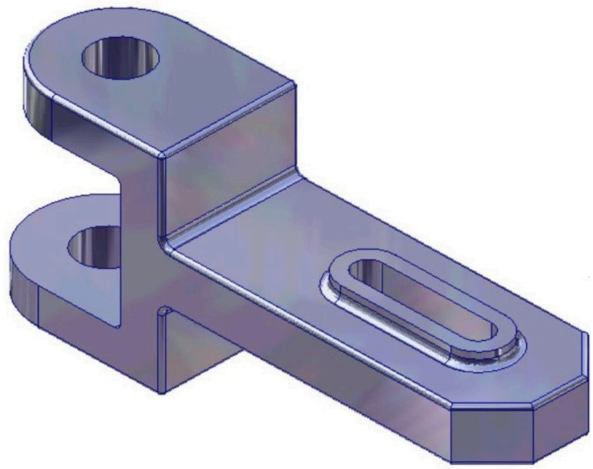


Figure Step 5A
Solid Model – Home View

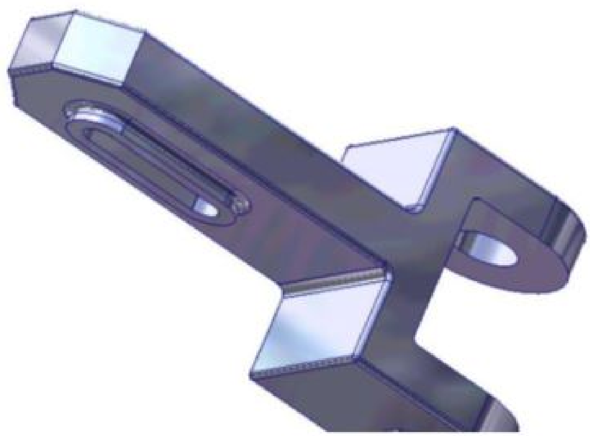


Figure Step 5B
Solid Model – Rotated View

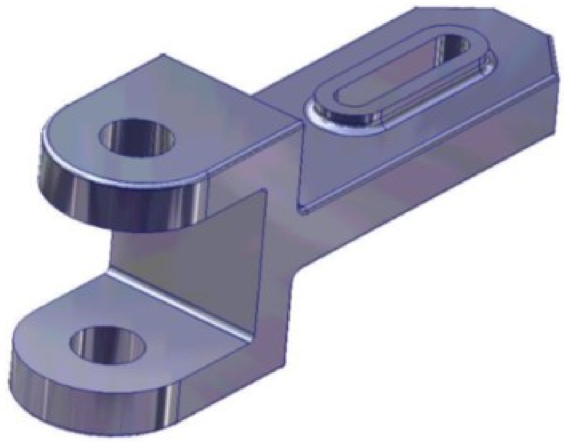


Figure Step 5C
Solid Model – Rotated View

Lab Exercise 16-2

Time allowed: 60 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab Lab 16-2	Inventor Course	Millimeters	Metric-Modules Part (mm).ipt	Silicon Nitrite – Polished	N/A

Step 1

Note the location of X0Y0Z0. Draw the base sketch on the Right Side view and revolve it to create the base model. (Figure Step 1A, 1B, 1C, and 1D)

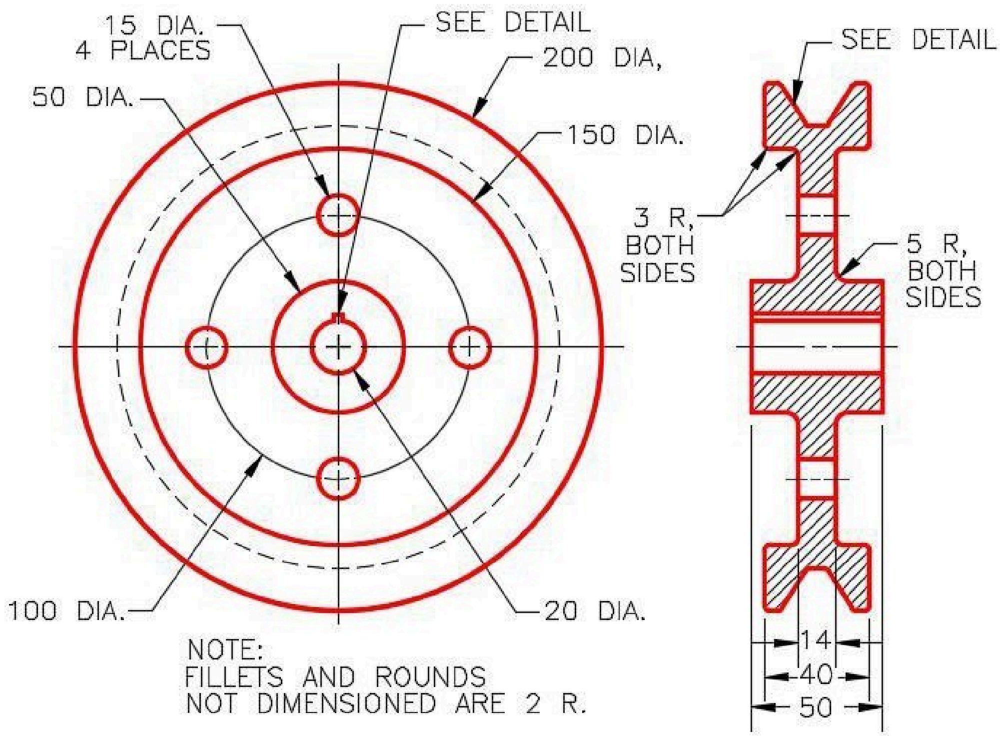


Figure Step 1A
Dimensioned Multiview Drawing [Click to see image full size]

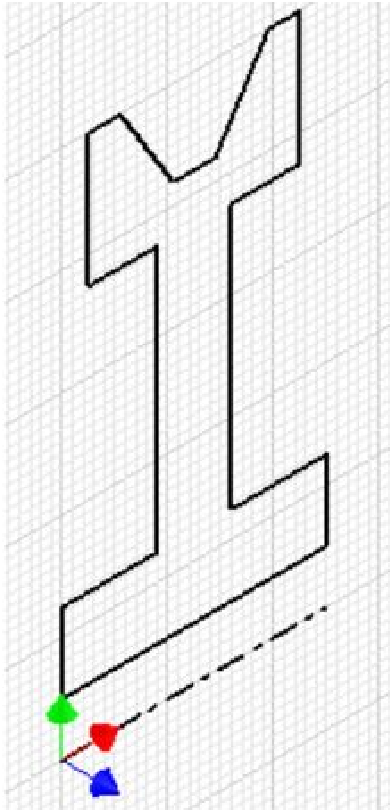


Figure Step 1B Suggested Base Sketch – Right Side View (YZ Plane)

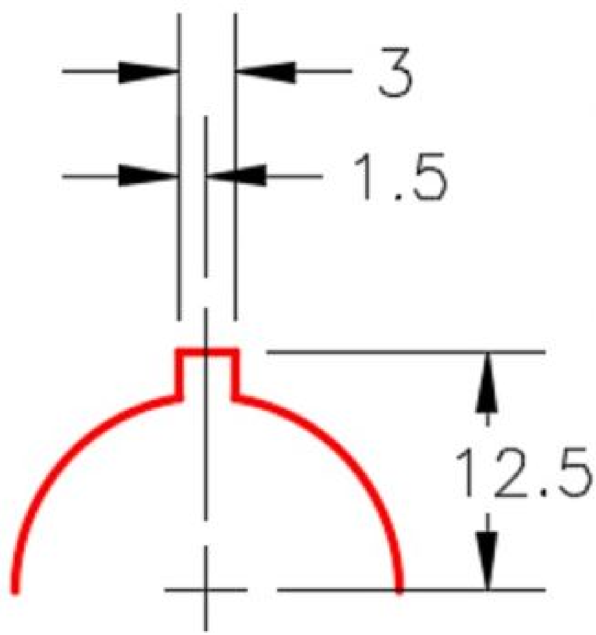


Figure Step 1C Keyway Detail

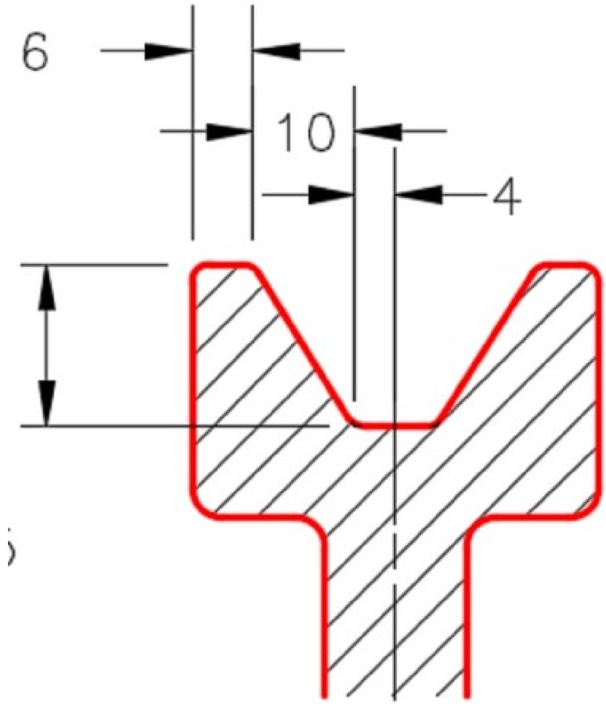


Figure Step 1D Section Detail

Step 2

Draw the necessary sketches and extrude them to complete the model.

Step 3

Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. Ensure that all objects, on all sketches, display purple on a black background.

Step 4

Create the fillets after the solid model is totally constructed.

Step 5

Apply the colour shown. (Figure Step 5A and 5B)

Figure Step 5A
Solid Model – Home View

Figure Step 5B
Solid Model – Rotated View

Part 4

Module 17 Angles

Learning Outcomes

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

1. Describe drawing inclined lines, aligned and angular dimensions, loops, trimming, and extending.
2. Apply the GENERAL DIMENSIONS command to insert aligned and angular dimensions on a sketch.
3. Apply the TRIM and EXTEND commands to trim and extend objects in a sketch.

Drafting Lesson: Auxiliary Views

When a model has an inclined side, its plane is not parallel to the horizontal and vertical sides of the glass box. If the inclined view is drawn in one of the predefined views in a multiview drawing, some or all parts of the object will not be their true size and shape. To correct this, an auxiliary view is drawn instead of a predefined view. An *auxiliary* view is a view looking perpendicular to the inclined plane as shown in Figure 17-1.

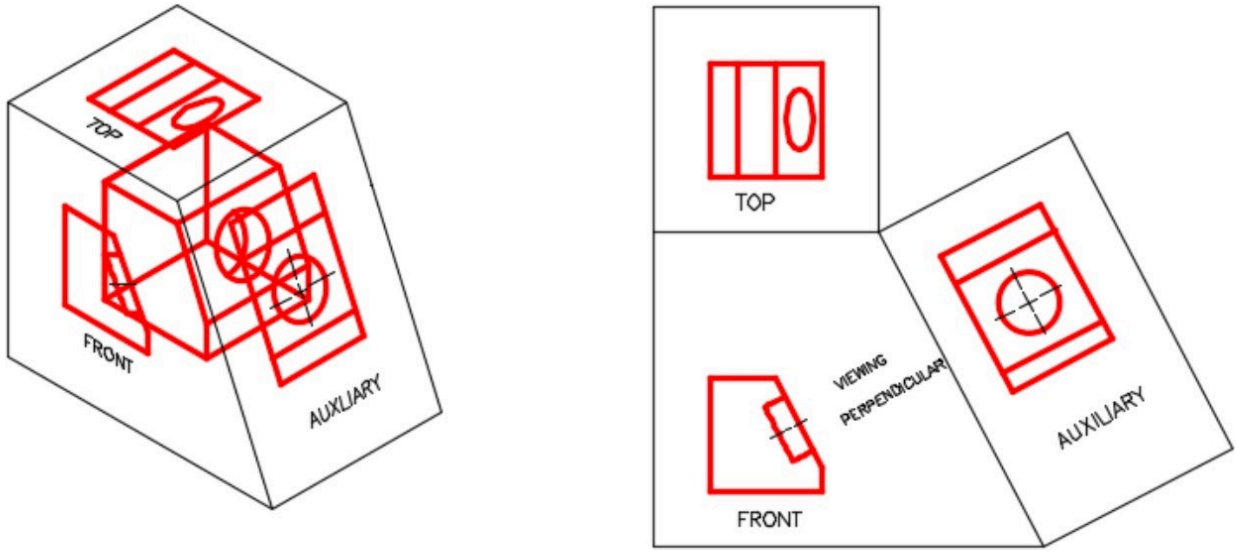


Figure 17-1
An Auxiliary View [Click to see image full size]

Drafting Lesson: Broken Views and Break Lines

To simplify or speed up drawing some of the views of a multiview drawing are only partially drawn. In these cases, the *cutoff* (sometimes called the *broken)* portion of the view is not required for the reader to visualize the object. Auxiliary views are frequently cutoff. When a view is cutoff, a *break line* is drawn to indicate where the view was broken as shown in Figure 17-2. A short break line and a long break line are drawn differently as show in Figure 17-2.

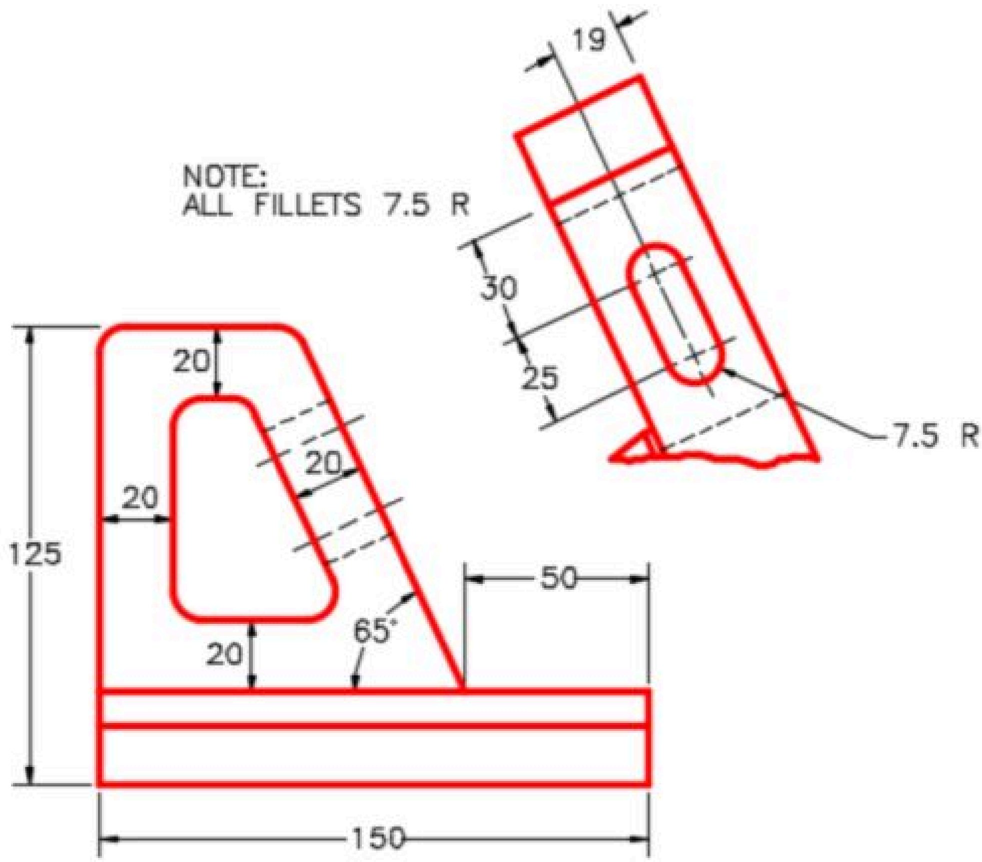
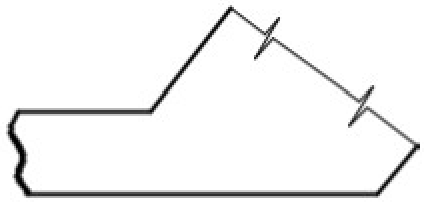


Figure 17-2
Broken Views and Break Lines

Short Break

Long Break

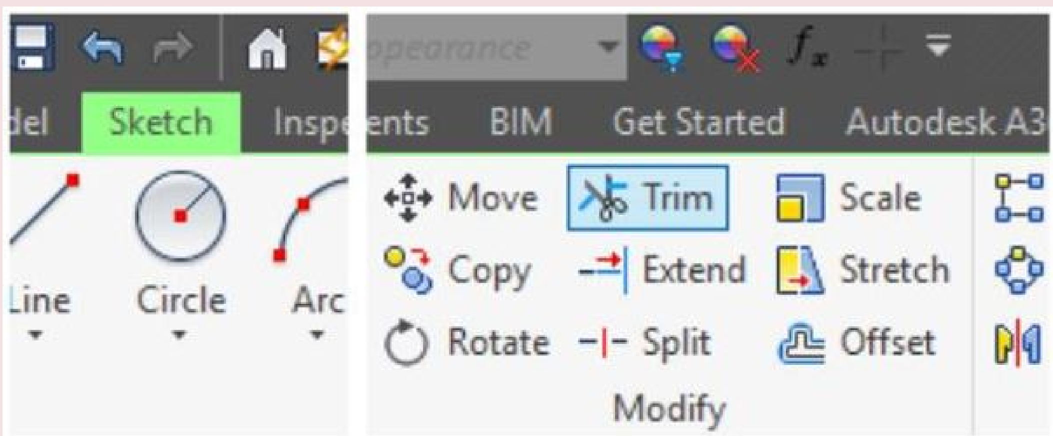


An Example of Using Break Lines

Inventor Command: TRIM

The TRIM command is used to trim a portion of an existing line or arc. The object to be trimmed must intersect an existing object. If it does not intersect an object, the complete object will be deleted instead of being trimmed.

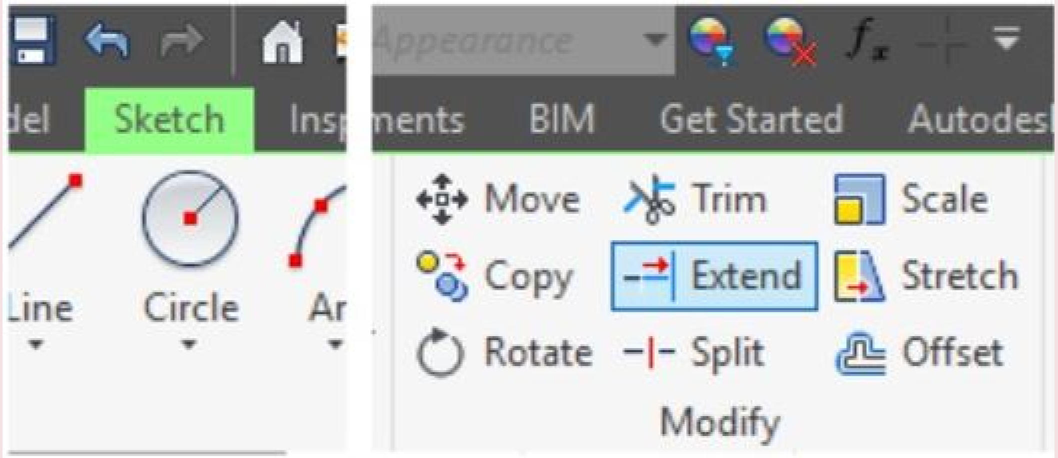
Shortcut: **X**



Inventor Command: EXTEND

The TRIM command is used to trim a portion of an existing line or arc. The object to be trimmed must intersect an existing object. If it does not intersect an object, the complete object will be deleted instead of being trimmed.

Shortcut: **X**



Drawing and Dimensioning Inclined Lines

Drawing and dimensioning inclined lines in sketches is a simple operation in Inventor compared to most CAD systems. The reason for this is that you can guess at the angle when drawing the inclined line rather then entering the exact number of degrees. After the sketch is complete, the angle is dimensioned using the exact angle and Inventor will adjust the sketch to match.

Aligned Dimensions

An *aligned* dimension is a dimension measuring the true length of a line or the true distance between two points. See Figure 17-3. The extension lines will be perpendicular and the dimension line will be parallel to the line or an imaginary line between two points.

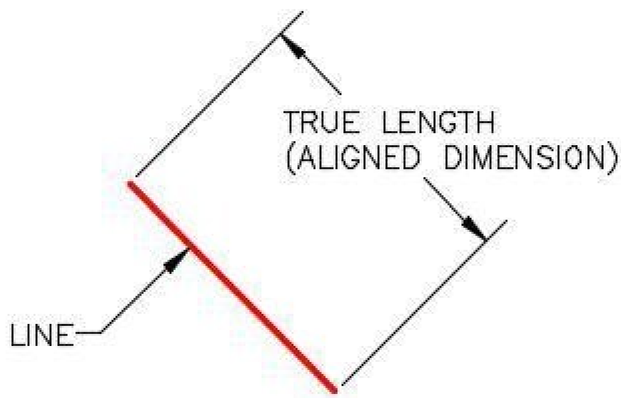


Figure 17-3
An Aligned Dimension

Placing an Aligned Dimension

To place an aligned dimension, enter the GENERAL DIMENSION command or the shortcut D and regardless if you are selecting a line, two points, or two lines to dimension, the same Aligned dimension icon will display as shown in Figures 17-4 and Figure 17-5.

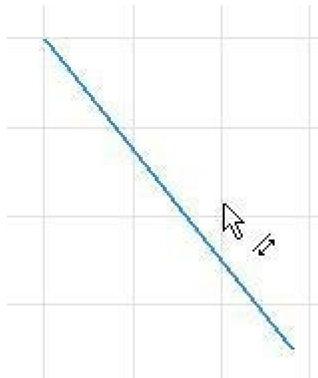


Figure 17-4 Placing a Aligned Dimension

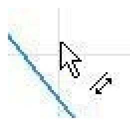


Figure 17-5 Aligned Dimension Icon

Angular Dimensions

An *angular* dimension is a dimension measuring the angle between two lines or the angle between the imaginary lines between three points. See Figure 17-6. The lines cannot be parallel to each other.

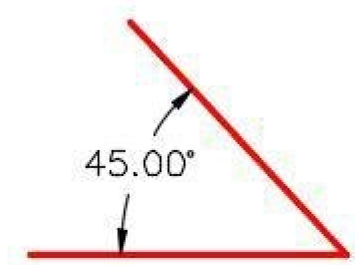


Figure 17-6 Angular Dimension

Placing an Angular Dimension

To place an angular dimension, enter the GENERAL DIMENSION command or the shortcut D and either select two lines or three points to place the angular dimension between.

The Two-Line Method

Select the first line. It will change colour. Move the cursor onto the second line and without selecting it, note how it changes colour. The Angular Dimension icon will display as shown in Figure 17-7. Select the second line. Drag the dimension to locate it. See Figure 17-8

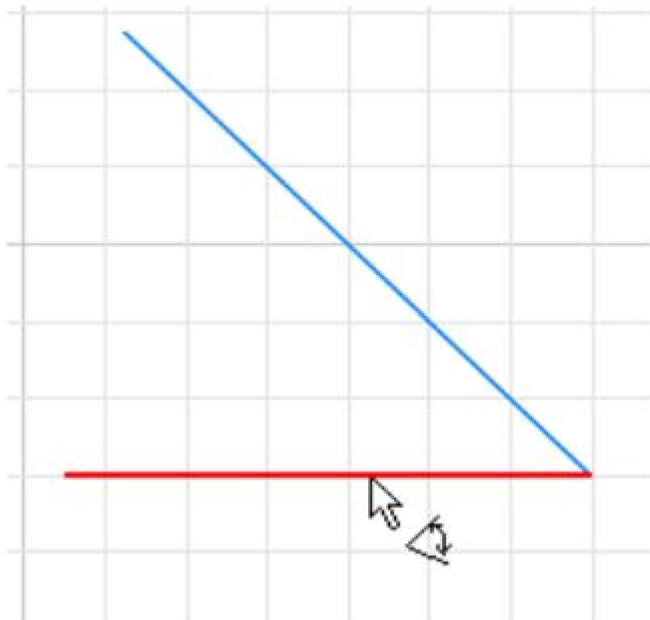


Figure 17-7
Placing an Angular Dimension – Two line Method

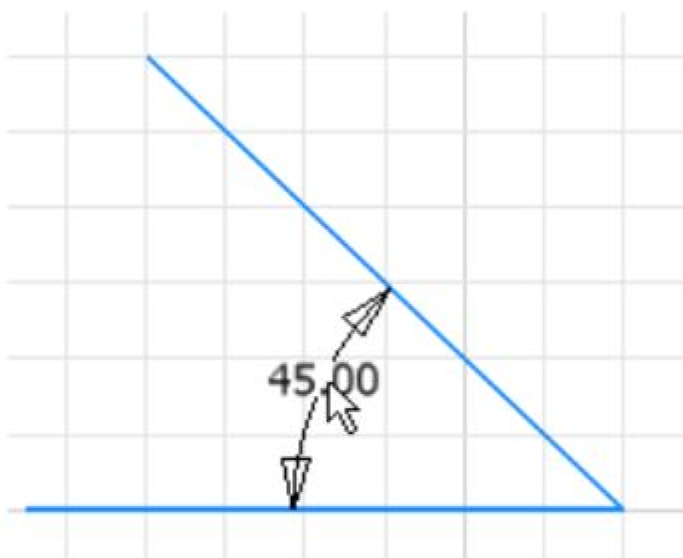


Figure 17-8
The Angular Dimension – Two line Method

The Three-Point Method

Select the first two points and move the cursor onto the third point as shown in Figure 17-9. The second point MUST be the vertex of the angle. The Angular Dimension icon will display as shown in Figure 17-9. Select the third point and drag the angular dimension to the desired location. See Figure 17-10.

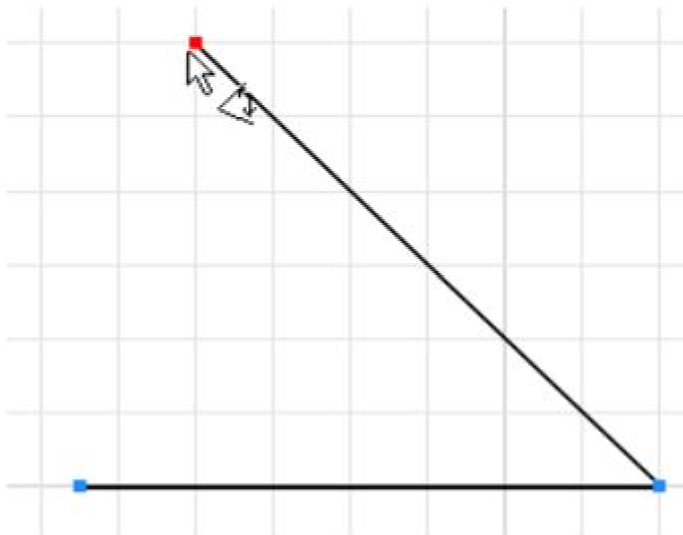


Figure 17-9
Placing an Angular Dimension – Three Point Method

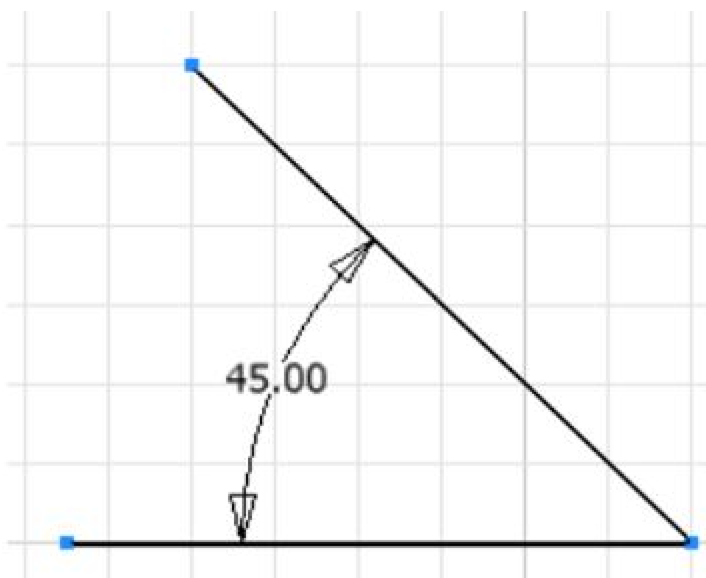


Figure 17-10
The Angular Dimension – Three Point Method

Geometrical Constraint Symbols			
Constraint	**Symbol**	**Icon**	**Definition**
Tangent			Constrains two objects tangent to each other.

WORK ALONG: Drawing Models that Contain Inclined Lines

Step 1

Check the default project and if necessary, set it to Inventor Course.

Step 2

Using the NEW command start a new part file using the template: English-Modules Part (in).ipt.

Step 3

Save the file with the name: Inventor Workalong 17-1. (Figure Step 3A, 3B, and 3C)

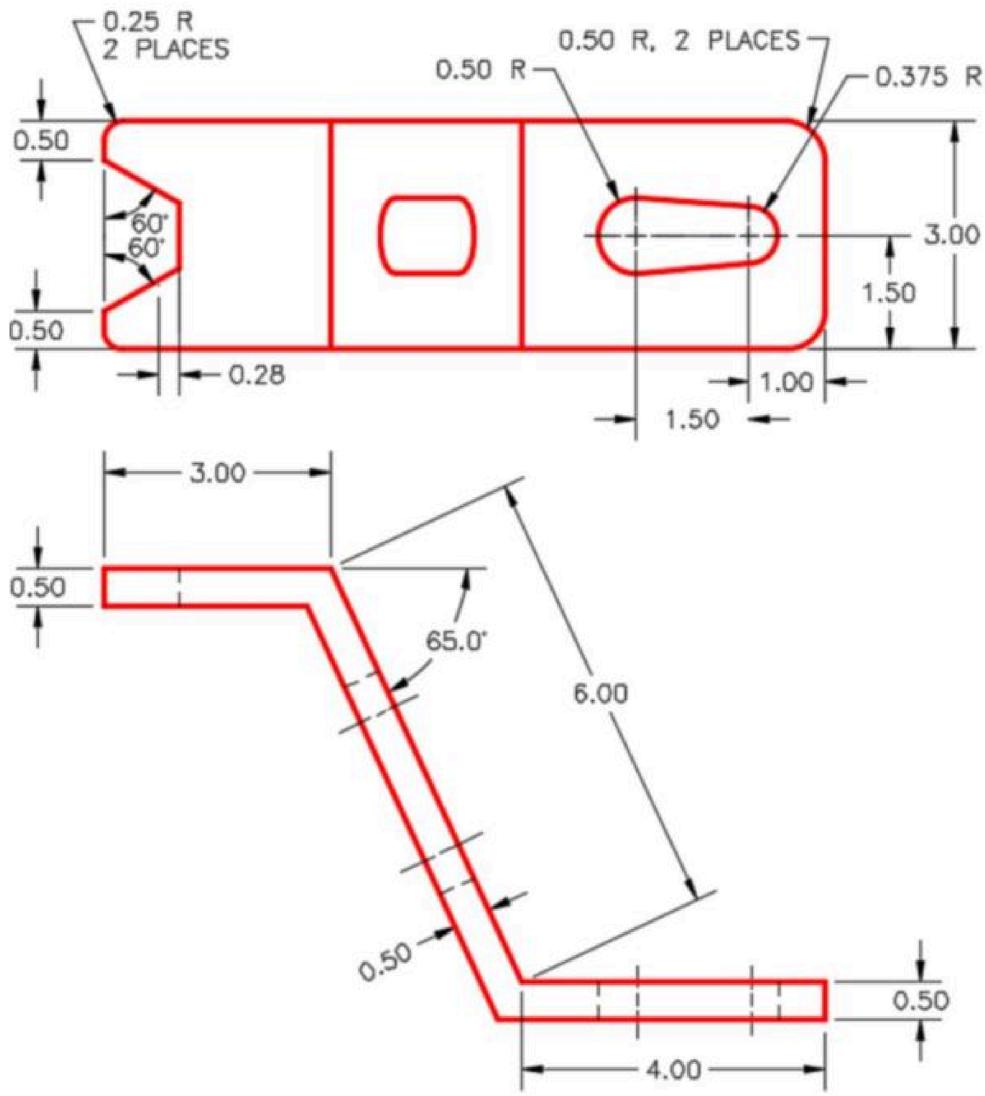


Figure Step 3A
Dimensioned Multiview Drawing [Click to see image full size]

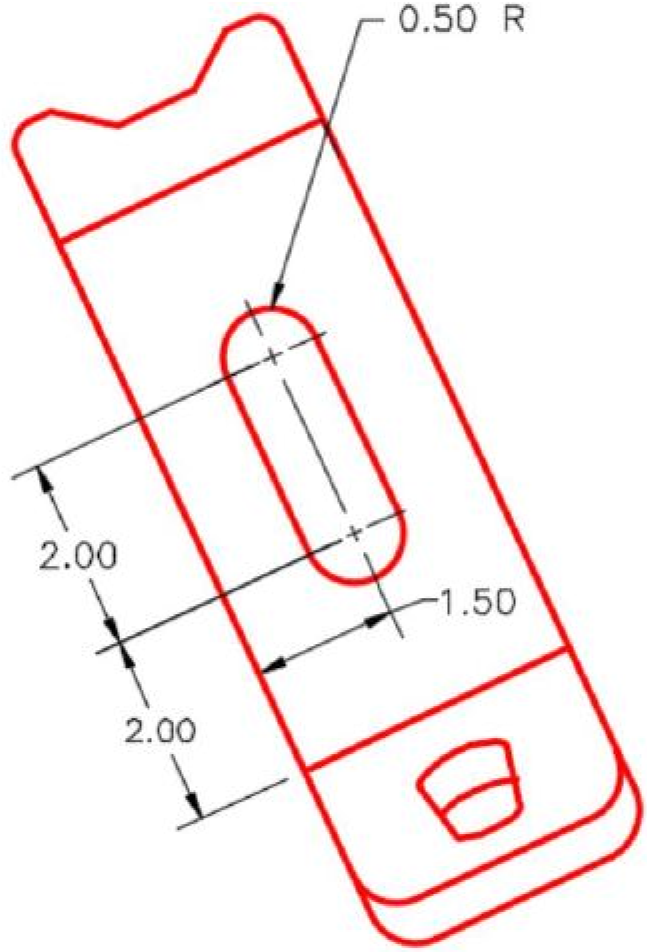


Figure Step 3B
Auxiliary View

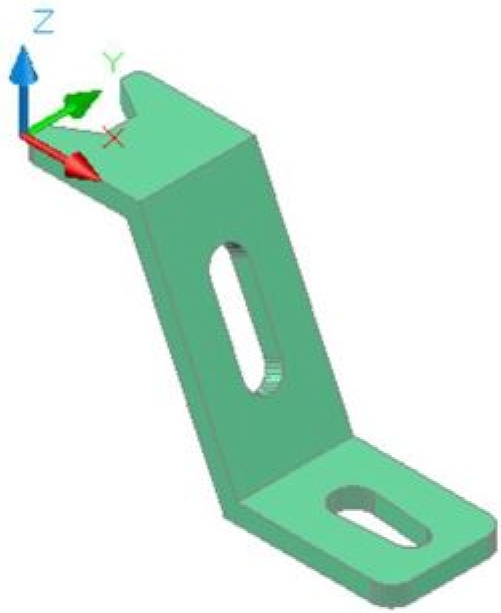


Figure Step 3C
3D Model
Home View

Step 4

Start the Base sketch on the Front or XZ plane.

Step 5

Project the Center Point onto the sketch.

Step 6

Draw the three top lines of the Front view and dimension them. Ensure that the sketch is fully constrained. (Figure Step 6)

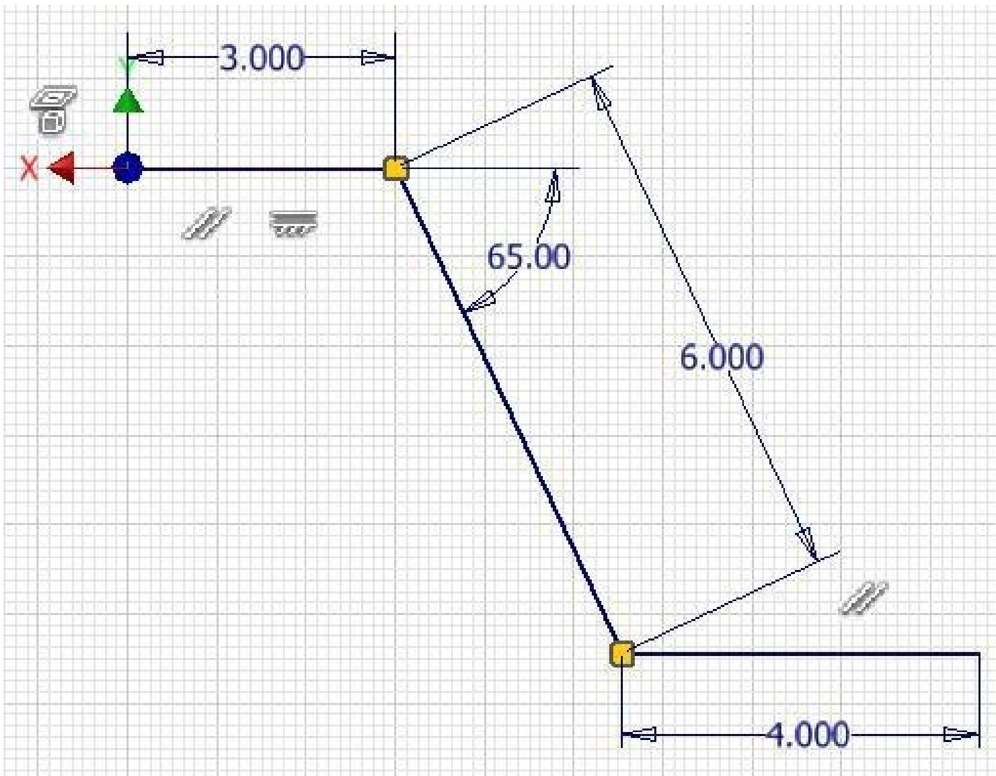


Figure Step 6

Step 7

Enter the OFFSET command. When prompted, select the top line. (Figure Step 7)

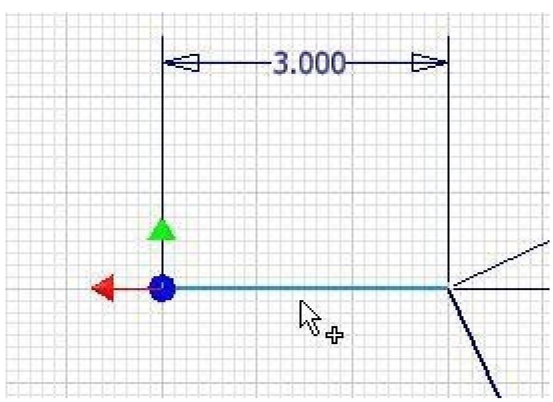


Figure Step 7

Step 8

Right click the mouse and in the Right-click menu, select Continue. Move the cursor down about 0.5 inches. The offset line will drag with it. Click to select the location. (Figure Step 8)

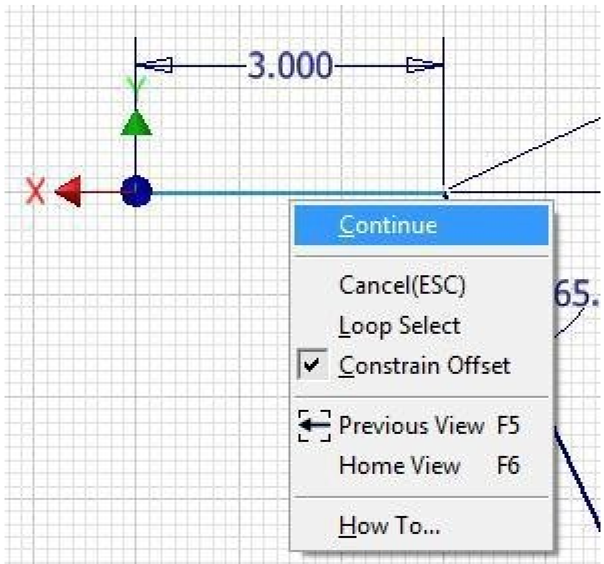


Figure Step 8

Step 9

Do the same for the other two lines. (Figure Step 9)

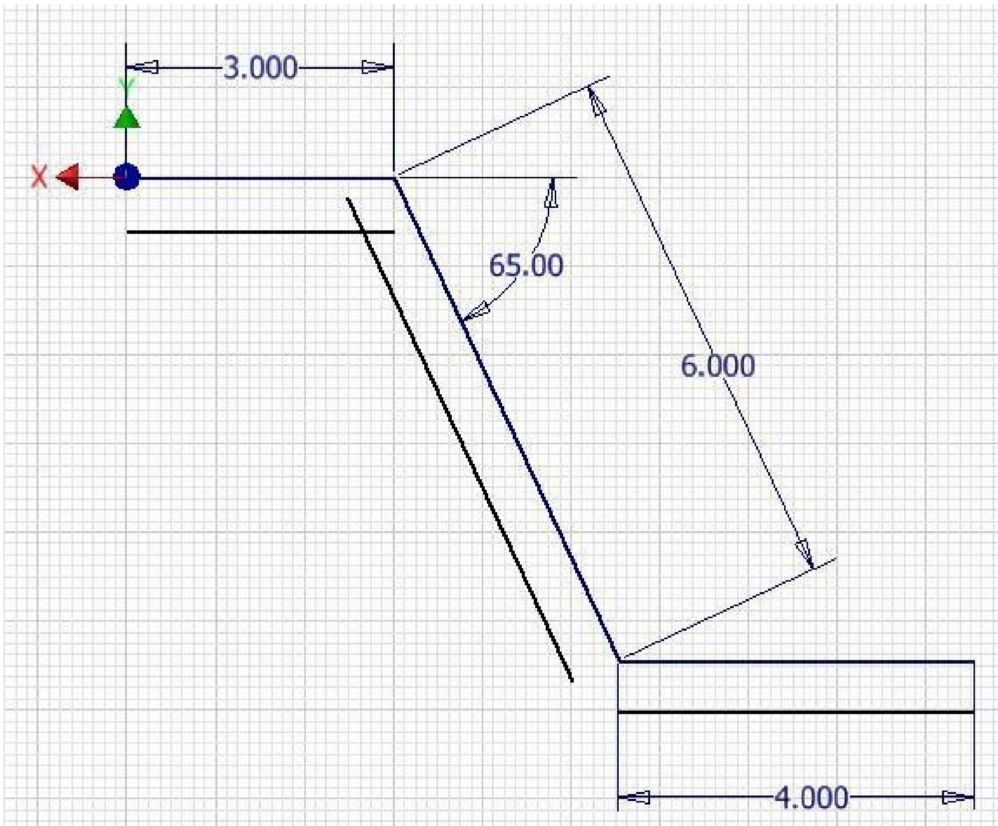


Figure Step 9

Step 10

Enter the TRIM command. When prompted, select the overlapping end of the lines on the top intersection. (Figure Step 10A and 10B)

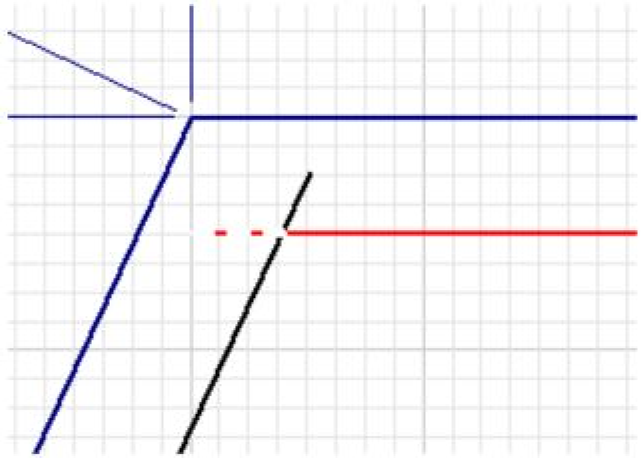


Figure Step 10A

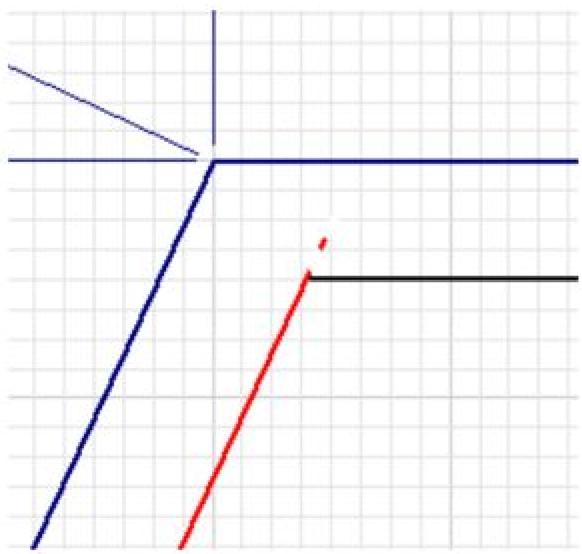


Figure Step 10B

Step 11

Enter the EXTEND command and extend the lines at the bottom intersection by selecting each of them. (Figure Step 11A and 11B)

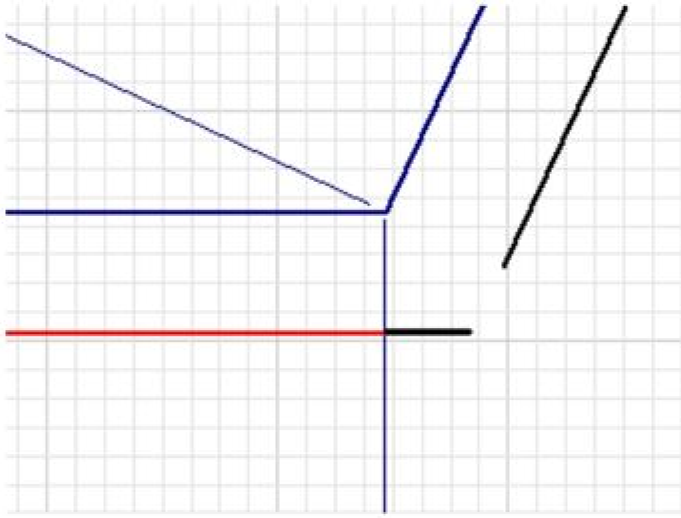


Figure Step 11A

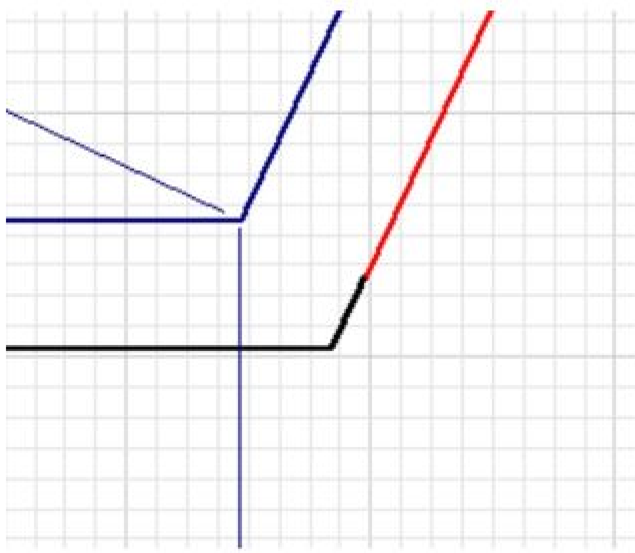
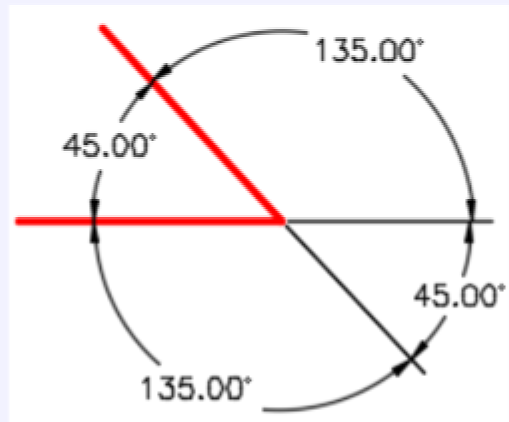


Figure Step 11B

MUST KNOW: When inserting an angular dimension, only one dimension can be placed at a time even though there is a choice of placing the dimension in four different locations and two different angles. The figure on the right shows the four different angular dimension locations and the two different angles that can be inserted.



Step 12

Add three dimensions for the 0.5 thickness. The sketch should be fully constrained. (Figure Step 12)

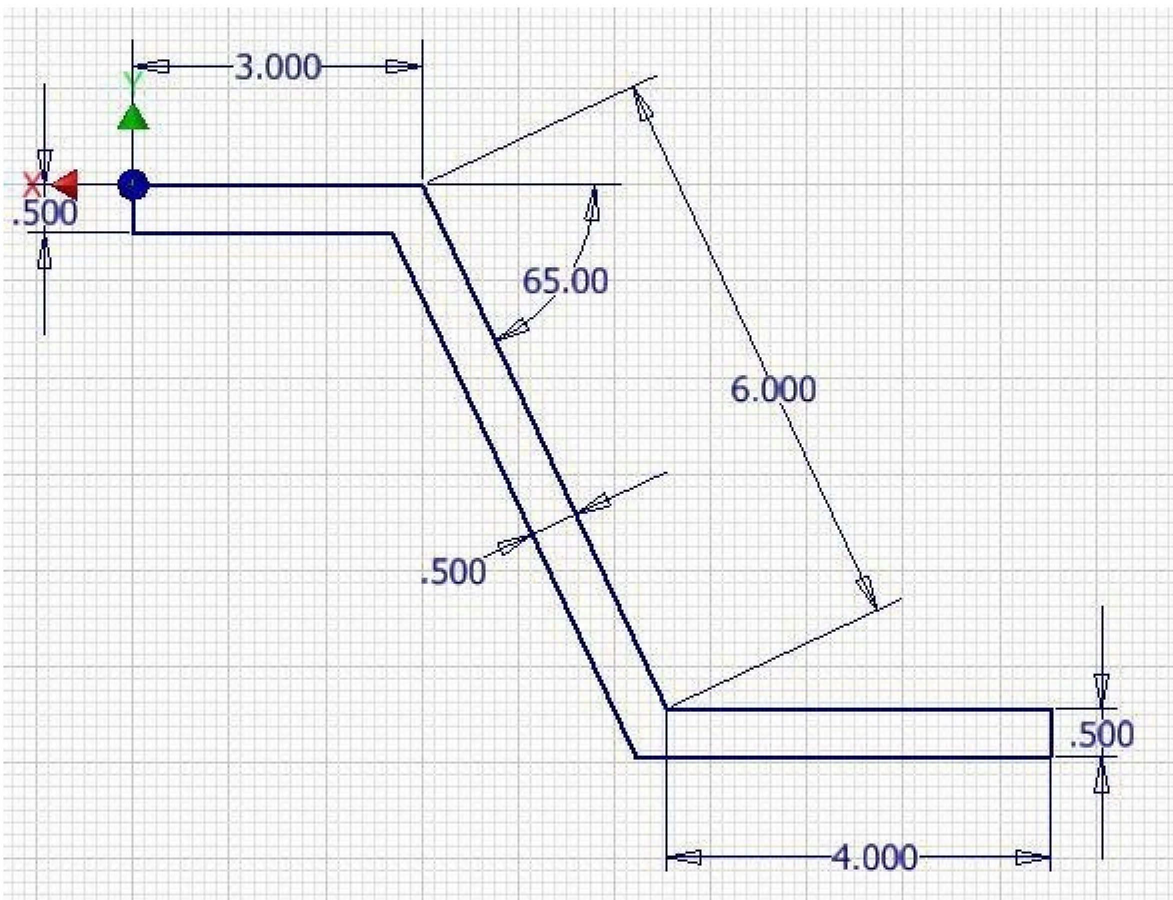


Figure Step 12

Step 13

Press F6 to return to Home view.

Step 14

Extrude the sketch. (Figure Step 14)

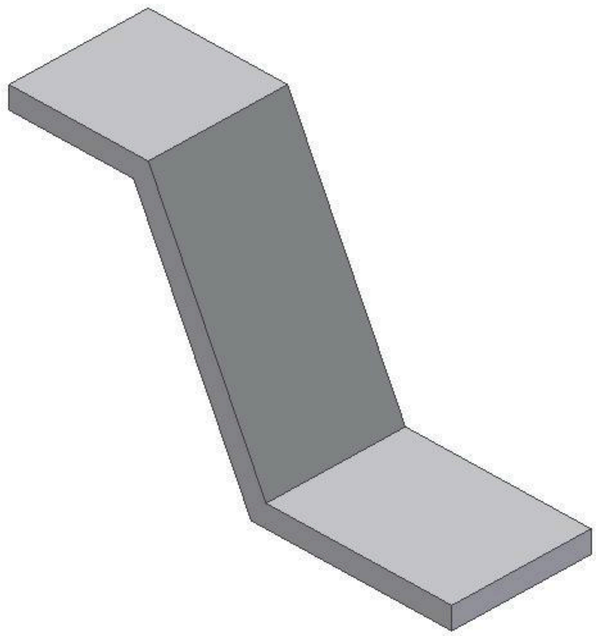


Figure Step 14

Step 15

Start a new sketch on the top plane. Draw three lines and add the dimensions to fully constrain the sketch. (Figure Step 15)

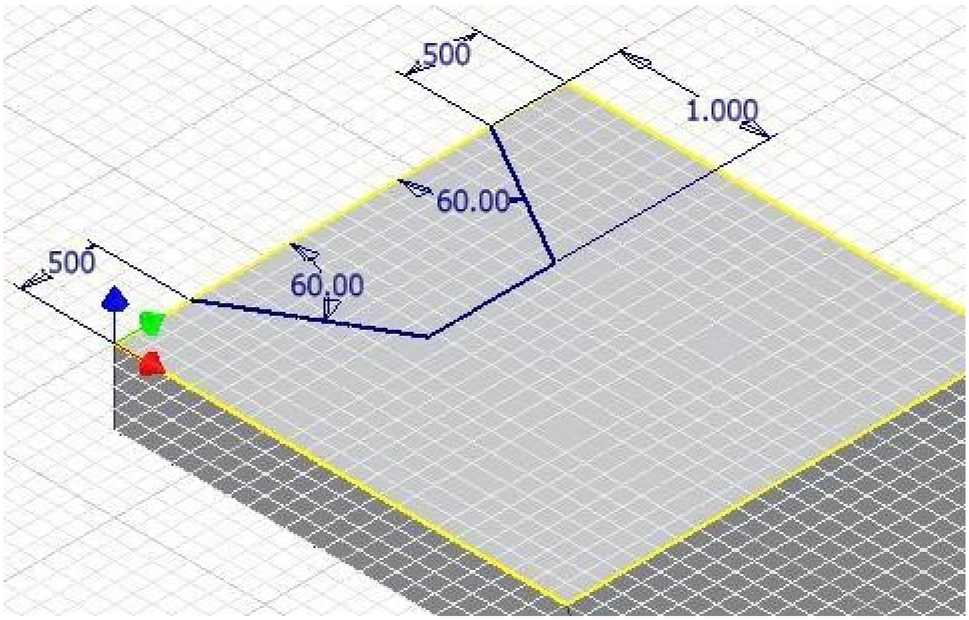


Figure Step 15

Step 16

Extrude the sketch using the cut option. (Figure Step 16)

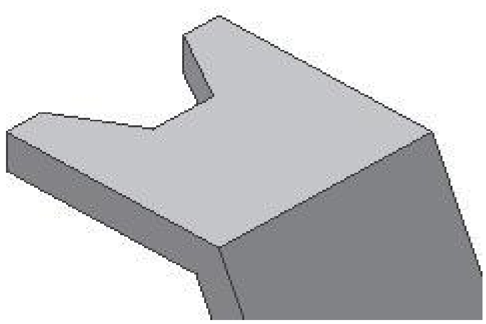


Figure Step 16

Step 17

Start a new sketch on the inclined plane. Draw three construction lines and dimension them to locate the centre of the circles. Ensure that the lines are fully constrained. (Figure Step 17)

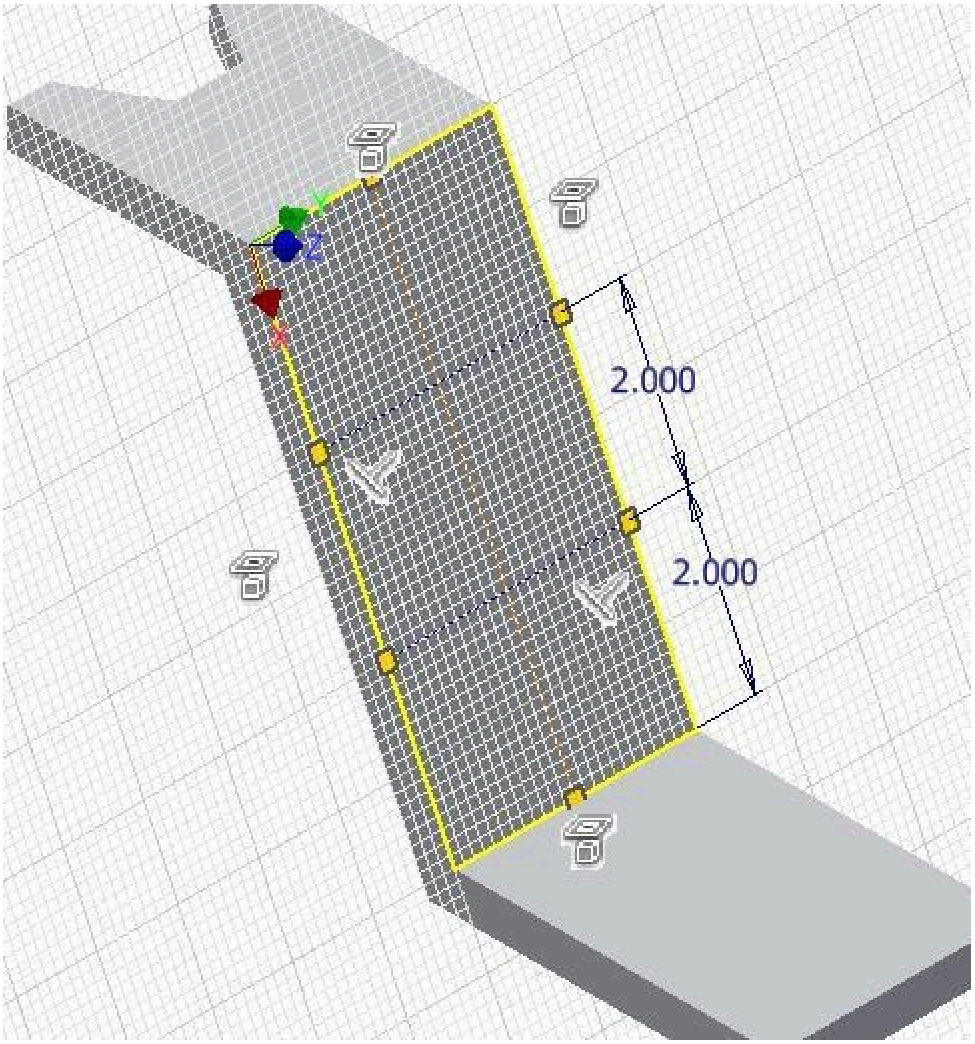


Figure Step 17

AUTHOR'S COMMENTS: Draw the vertical line from the midpoint of the top edge to the midpoint of the bottom edge. That way it will not have to be dimensioned. Draw the horizontal lines perpendicular to the vertical line or one of the edges and snap onto the edge lines.

Step 18

Insert two circles locating their centers at the intersection of the construction lines. Dimension only one of them and then apply the Equal constrain to the other circle. (Figure Step 18)

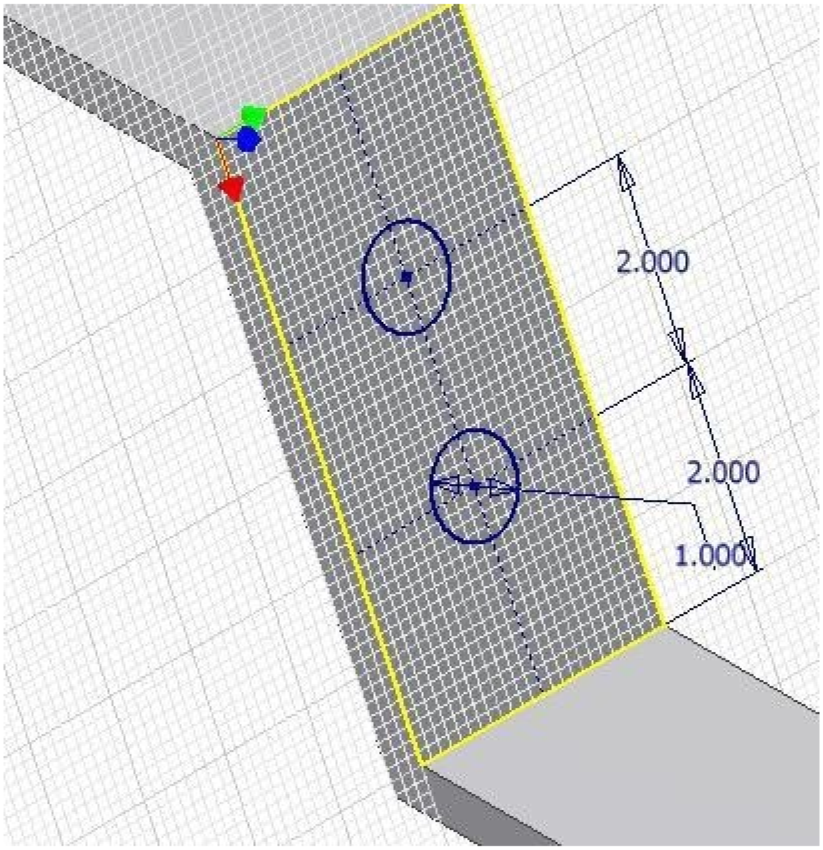


Figure Step 18

Step 19

Draw a line from one circle to the other. Don't worry about constraining them tangent at this time. Ensure that the Snap On icon appears when you select the endpoint of the lines. (Figure 19A and 19B)

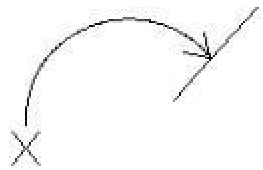


Figure Step 19A

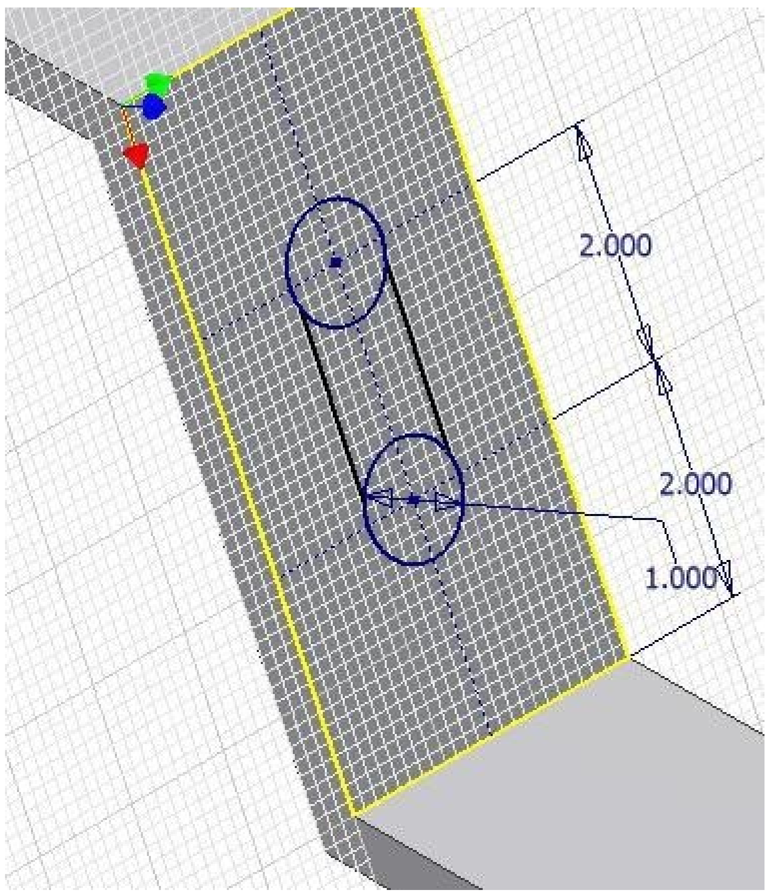


Figure Step 19B

Step 20

In the right-click menu, select Create Constrain – Tangent. (Figure Step 20).

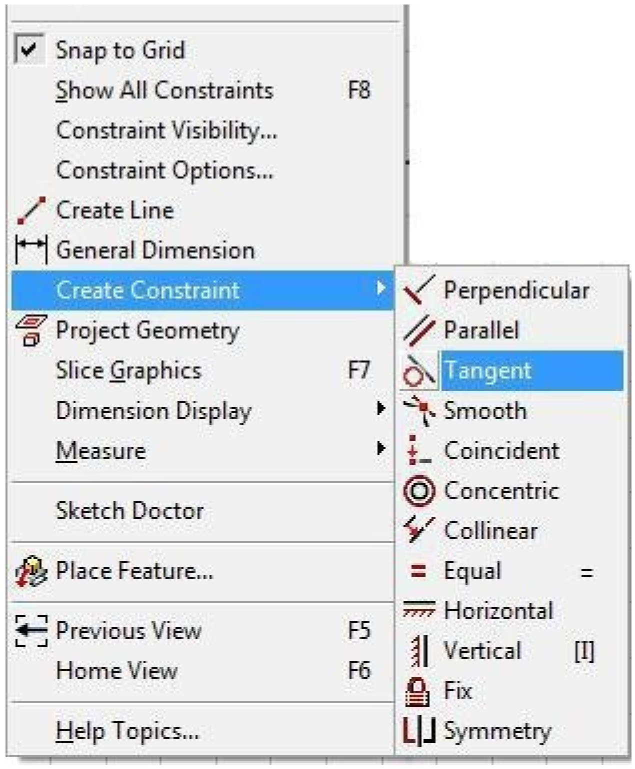


Figure Step 20

Step 21

Apply the Tangent constraint between the circle and the line. Repeat with the other circle. (Figure Step 21A and 21B)

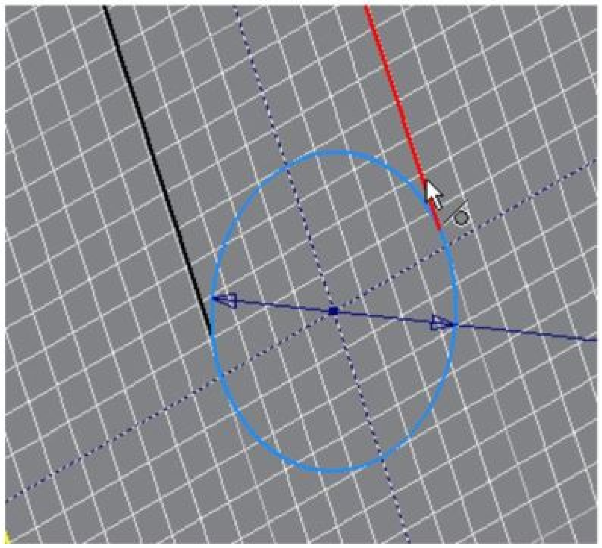


Figure Step 21A [Click to see image full size]

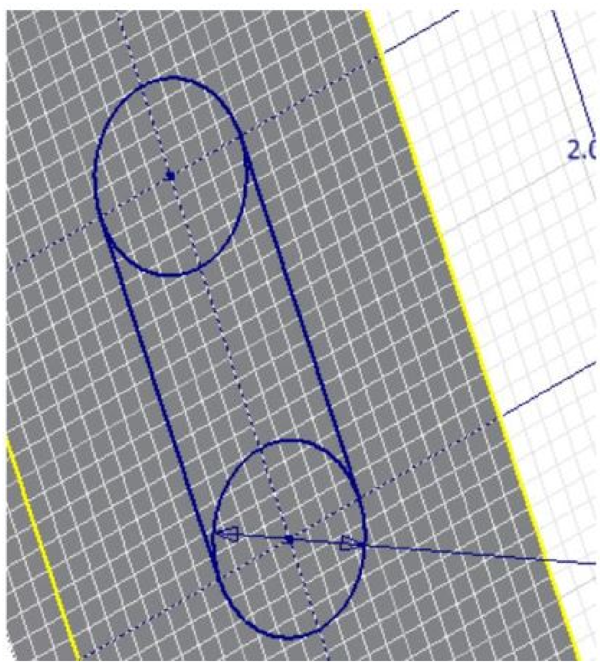


Figure Step 21B [Click to see image full size]

Step 22

Trim the circles. This will take four steps. (Figure Step 22)

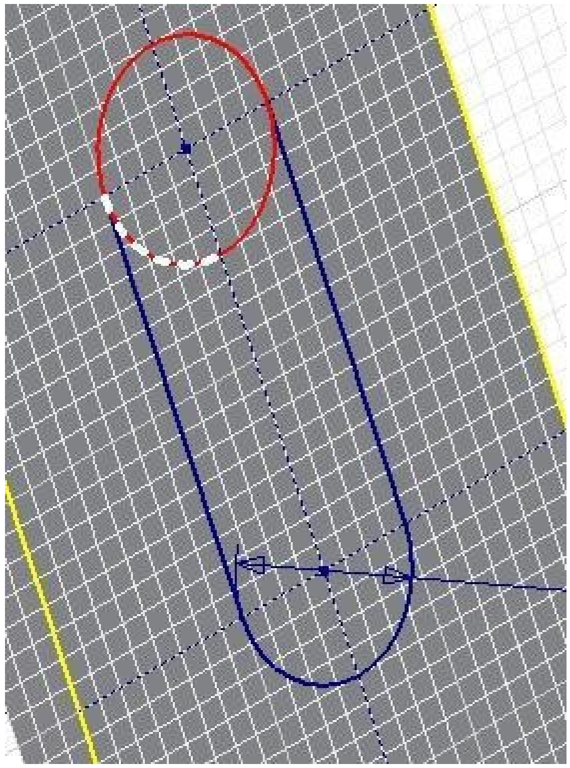


Figure Step 22

Step 23

Extrude the sketch. (Figure Step 23)

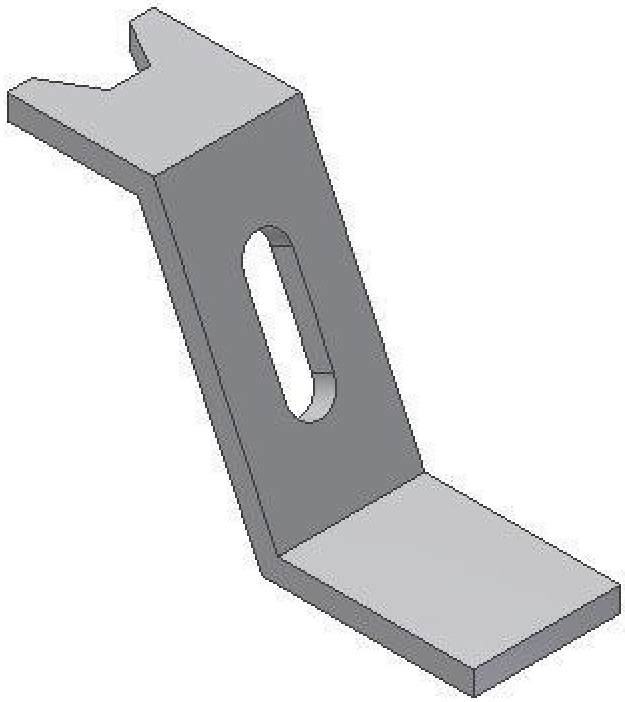


Figure Step 23

Step 24

Draw a 2D Sketch on the bottom plane. Using what you just learned, ensure that you constrain the lines tangent to the circles and then trim. (Figure Step 24A and 24B)

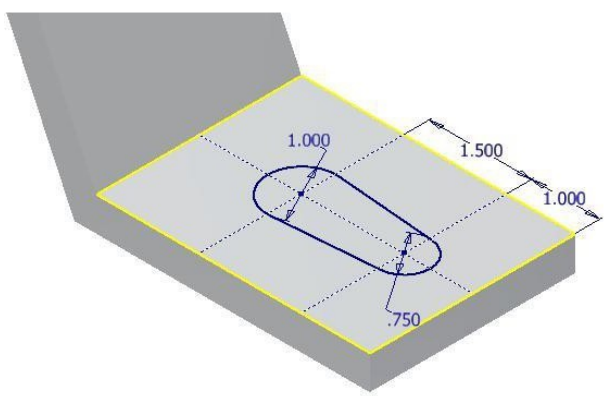


Figure Step 24A [Click to see image full size]

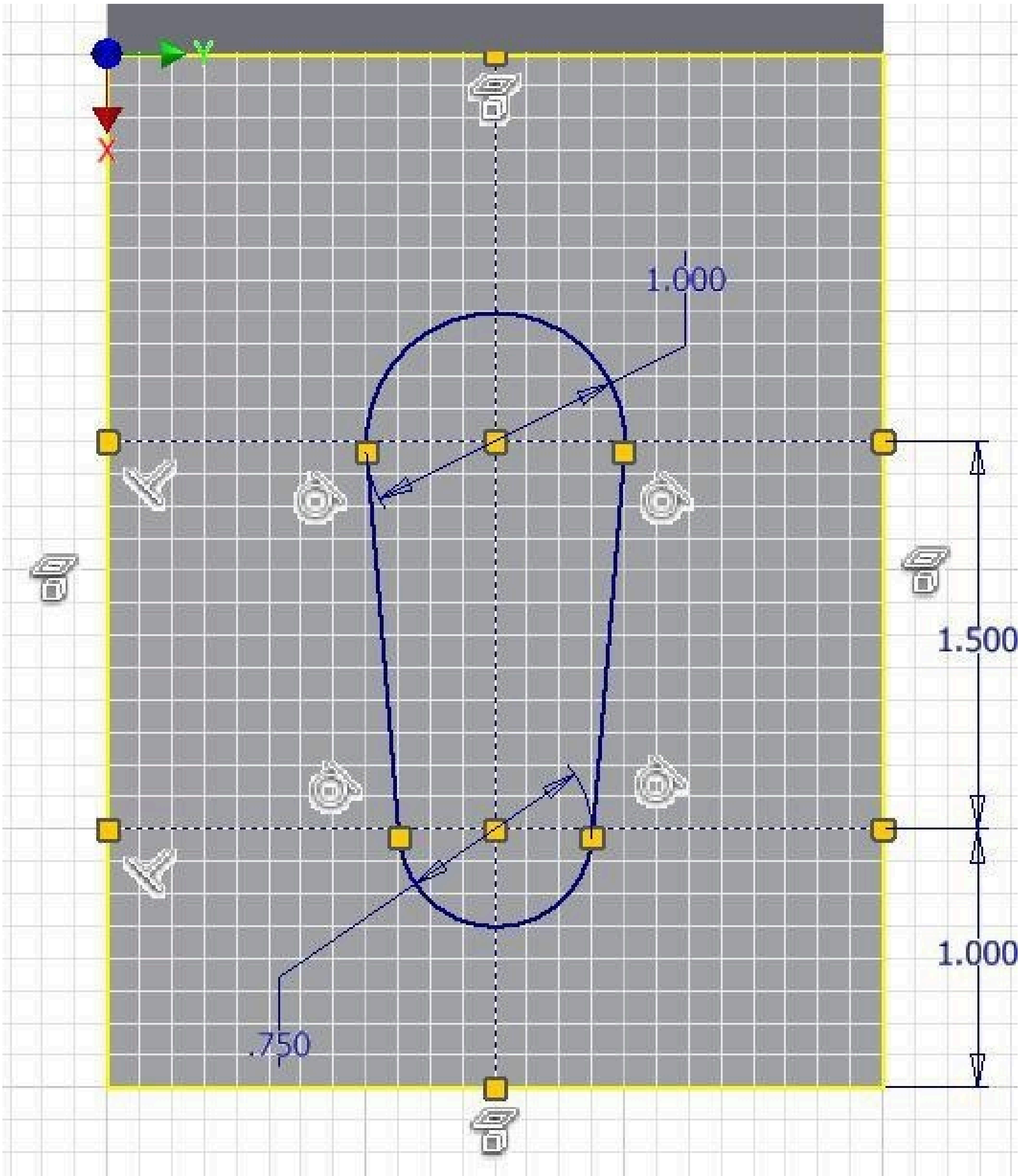


Figure Step 24B

Step 25

Extrude the sketch. (Figure Step 25)

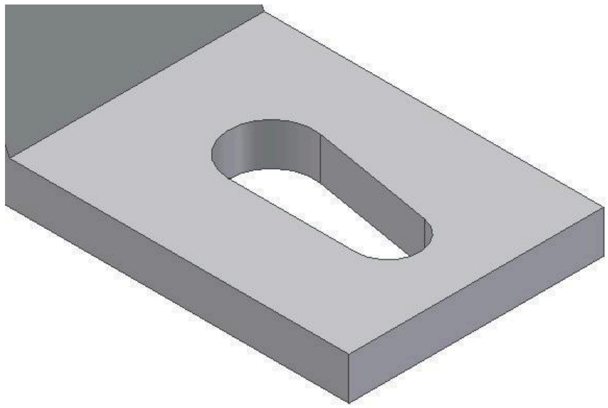


Figure Step 25

Step 26

Insert the fillets and change to the color: Orange to complete the solid model. (Figure Step 26)

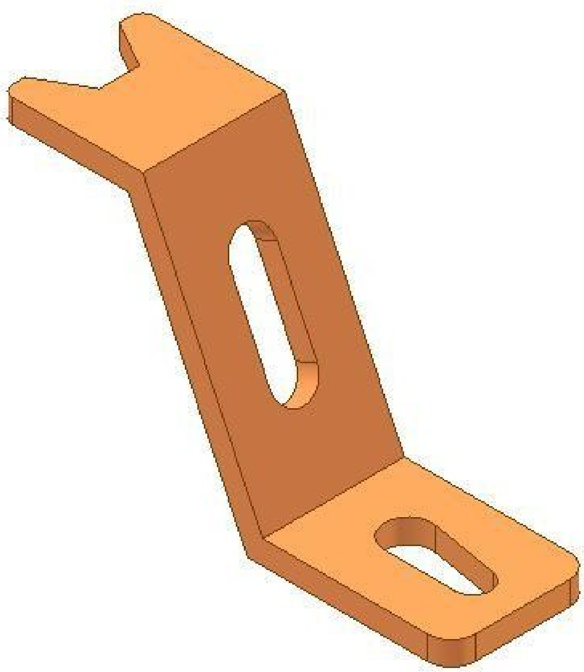
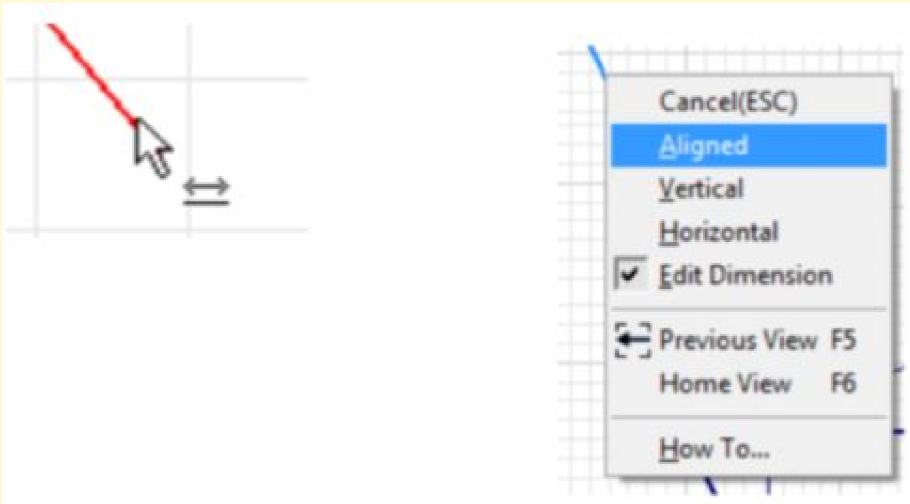


Figure Step 26

Step 27

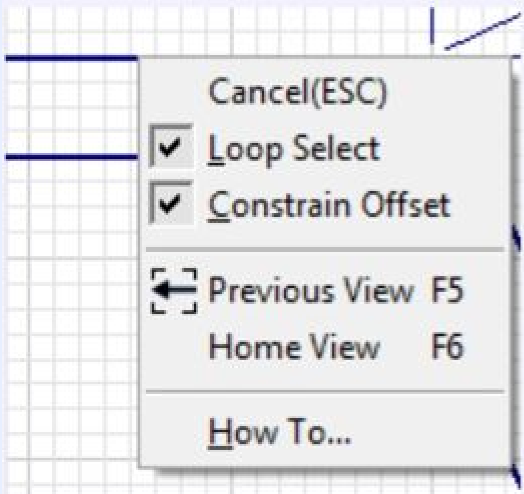
Save and close the part file.

USER TIP: When inserting an aligned dimension and the Linear dimension icon displays, as shown in the figure immediate right, rather then the Aligned dimension icon, you can change that and force Inventor to place an aligned dimension. Right-click the mouse while the icon is displayed. In the Right-click menu, select Aligned as shown in the figure far right. This will also work in reverse. If the Aligned dimension icon displays, you can instruct Inventor to place a linear dimension either horizontal or vertical.



USER TIP: The TRIM command can be used to completely delete an object rather then just trimming it. If the object to be deleted does not intersect another object, simply press X and select the object to be deleted. If it intersects another object, it will take you more picks to delete it, but, it is still possible. The reason that it is best to use the TRIM command rather then the DELETE command to delete objects is the fact that TRIM has a shortcut (X) while the DELETE command does not have a shortcut. Entering a shortcut on the keyboard is faster then clicking an icon.

MUST KNOW: When offsetting most objects, the offset object can be geometrically constrained to the existing object. If the angle of the object that was offset is modified or the object is moved, the object that was offset will maintain its position in relation to the offset object. In the Right-click menu, during the OFFSET command, the Constrain Offset can be enabled or disabled as required. See the figure on the right.



Key Principles

Key Principles in Module 17

1. You can guess at the angle when drawing inclined lines rather then entering the exact number of degrees. After the sketch is complete, the angles are dimensioned using the exact angle and Inventor will adjust the sketch to match.
2. The TRIM command is used to trim a portion of an existing line or arc. The object to be trimmed must intersect an existing object. If it does not intersect an object, the complete object will be deleted instead of being trimmed.
3. The EXTEND command is used to extend the length of an existing line or arc.

Lab Exercise 17-1

Time allowed: 60 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab 17-1	Inventor Course	Millimeters	Metric-Modules Part (mm).ipt	Beige	N/A

Step 1

Project the Center Point onto the base plane.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude or revolve them to produce the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to maintain the objects shape and size. (Figure Step 2A and 2B)

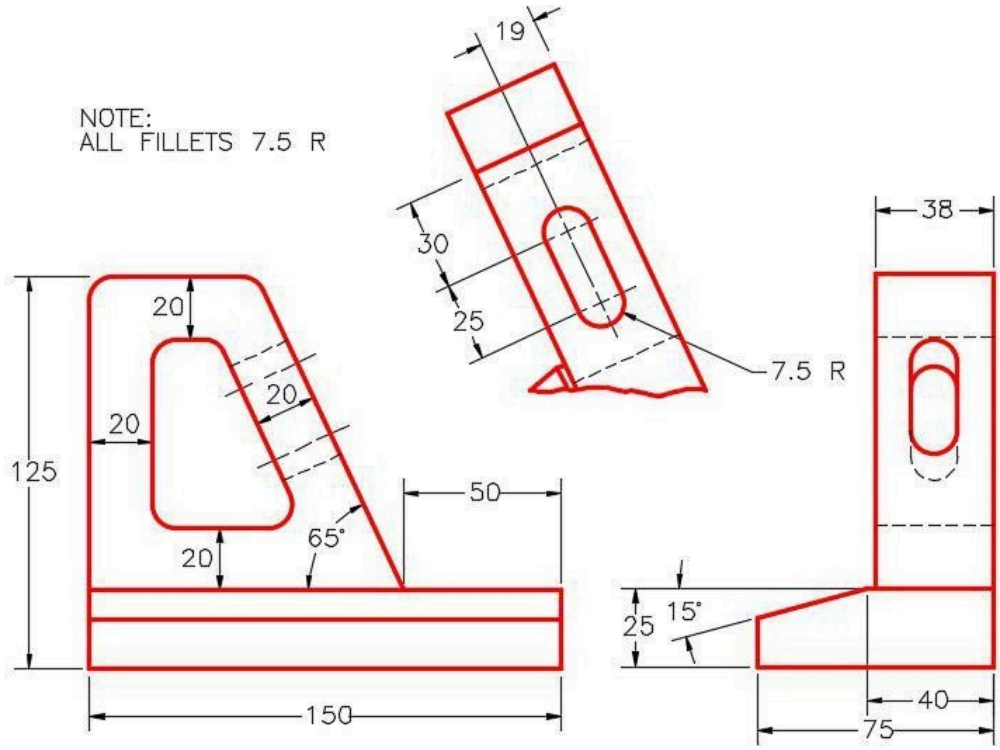


Figure Step 2A
Dimensioned Multiview Drawing [Click to see image full size]

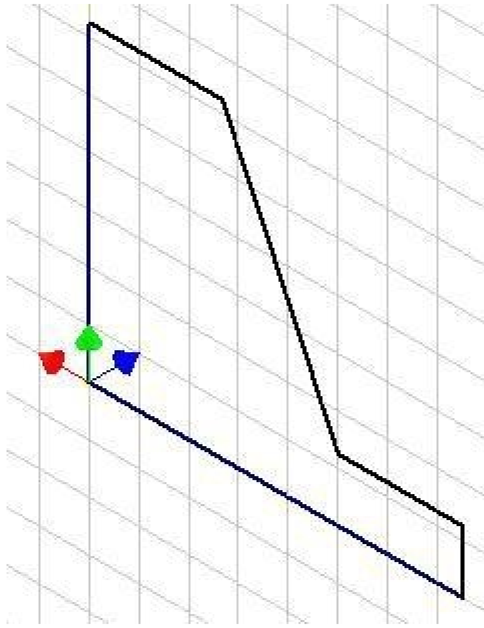
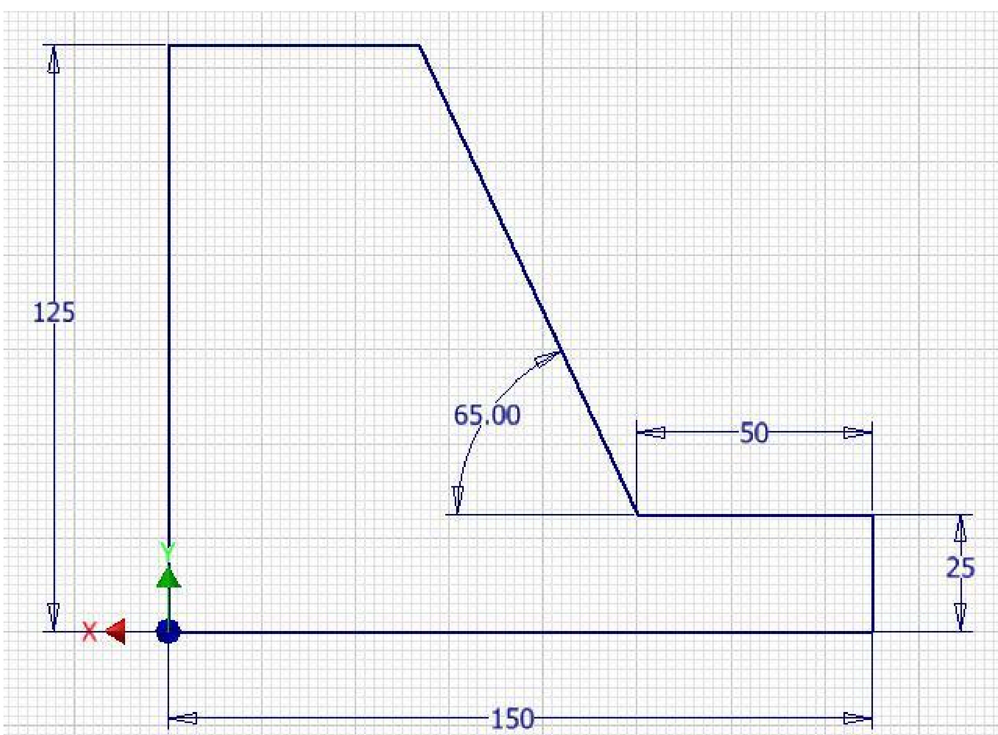


Figure Step 2B Suggested Base Sketch – Front (XZ) Plane

Step 3

Apply the colour shown above. (Figure Step 3)

Figure Step 3
Completed Solid Model
Home View



Author's Base Sketch [Click to see image full size]

Step 4

Create all fillets after the solid model is totally constructed.

Lab Exercise 17-2

Time allowed: 60 minutes.

Part Name	**Project**	**Units**	**Template**	**Color**	**Material**
Inventor Lab 17-2	Inventor Course	Inches	English-Modules Part (in).ipt	Nickle	N/A

Step 1

Project the Center Point onto the base plane.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude or revolve them to produce the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to maintain the objects shape and size. (Figure Step 2A and 2B)

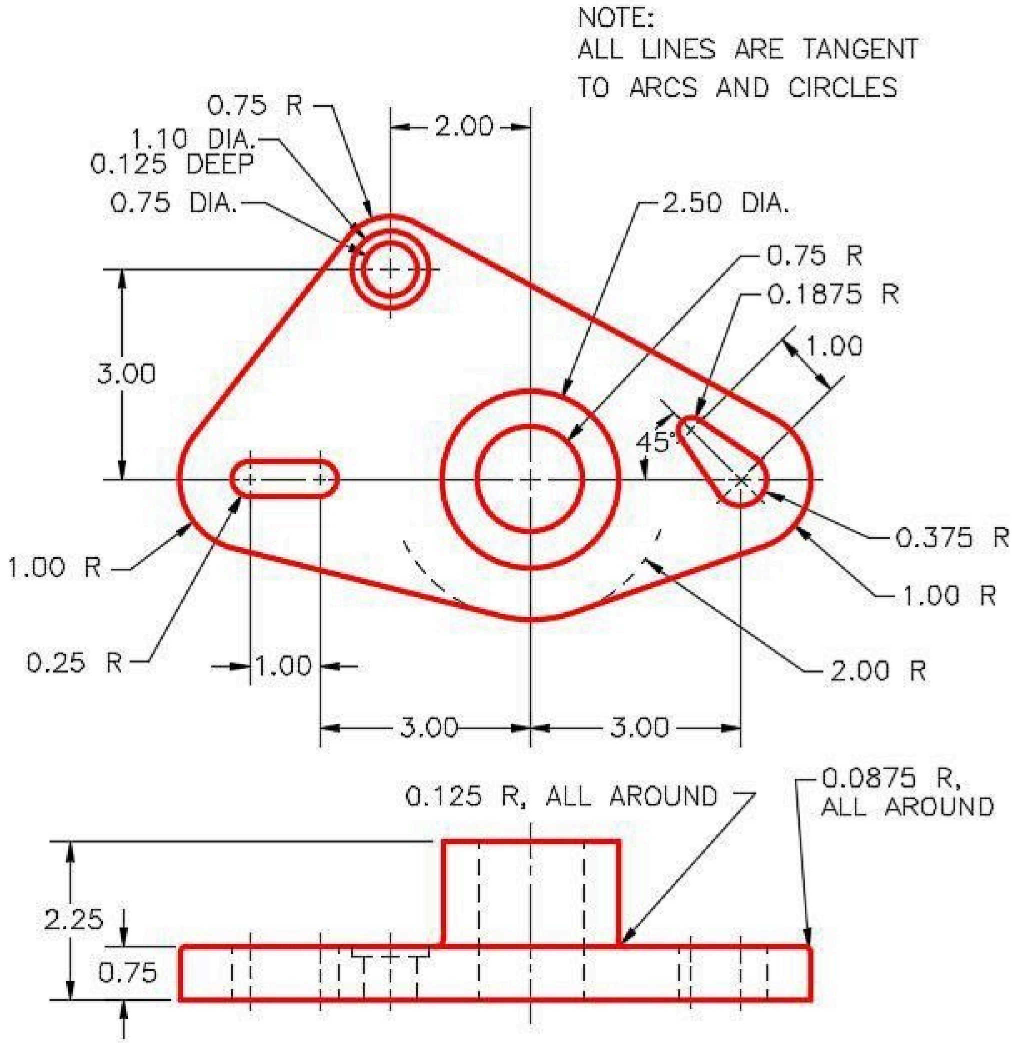


Figure Step 2A
Dimensioned Multiview Drawing [Click to see image full size]

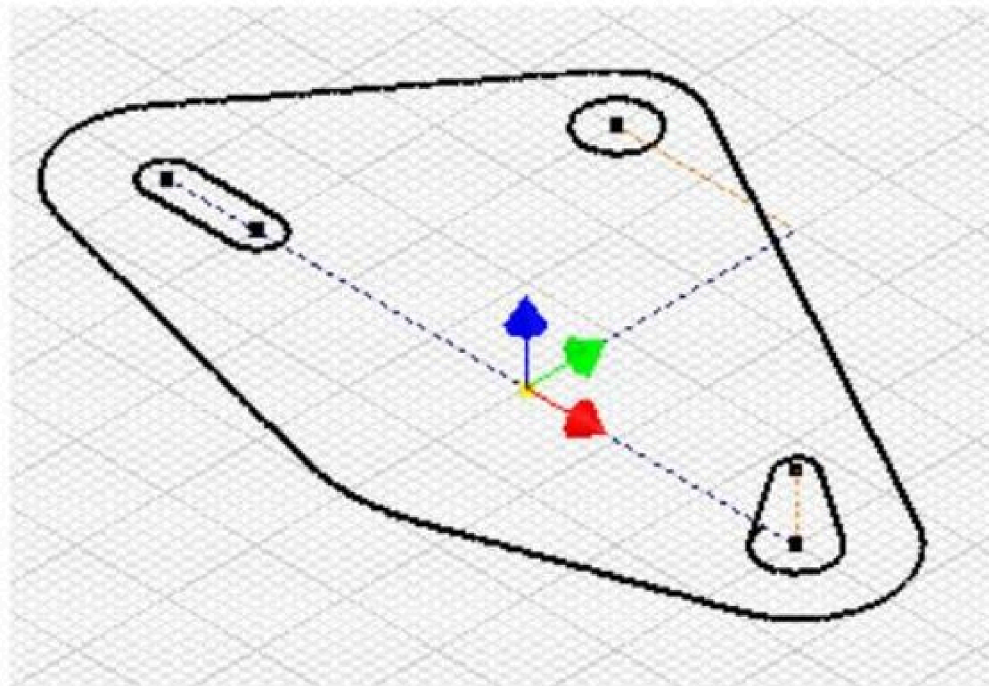


Figure Step 2B
Suggested Base Sketch –
Top (XY) Plane) [Click to see image full size]

Step 3

Apply the colour shown above. (Figure Step 3)

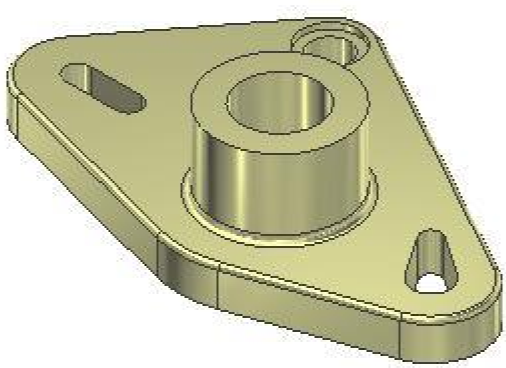
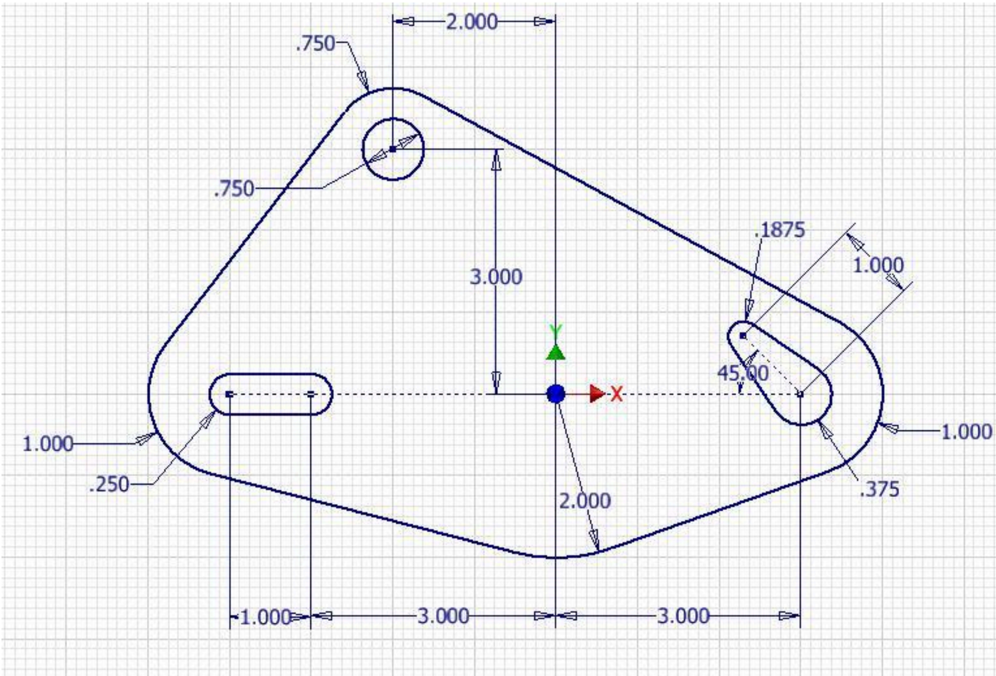


Figure Step 3
Completed Solid Model – Home View
Author's

Step 4

Create all fillets after the solid model is totally constructed.



Author's Base Sketch [Click to see image full size]

Module 18 Editing Geometry

Learning Outcomes

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

1. Describe how to select objects using windows and crossing windows.
2. Describe and apply the THREE POINT ARC, TWO POINT RECTANGLE, and COPY commands.

Methods of Selecting Objects

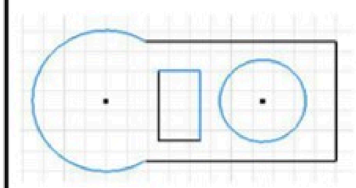
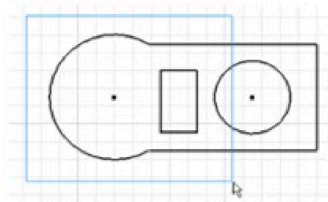
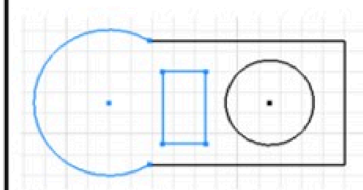
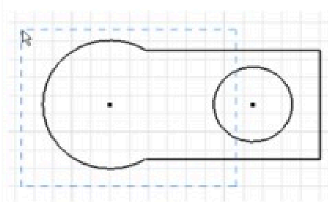
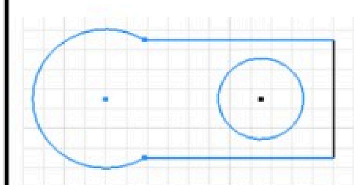
Methods of Selecting Objects			
Method	**Description**	**Window Display**	**Selected Object Display**
pick	Selects one object at a time using the graphic cursor. To pick more then one object at a time to create a selection set, hold down the SHIFT or CTRL keys while selecting the objects. Selected objects will change color.		
window	Selects all the objects that are totally inside a window defined by two user picks. A *window* will always appear as a solid line and is created by picking the first point on the left and moving right to pick the second point.		
cross	Selects all the objects that are totally inside and the ones that cross a window defined by two user picks. A *crossing window* will always appear as a dashed line and is created by picking the first point on the right and moving left to select the second point.		

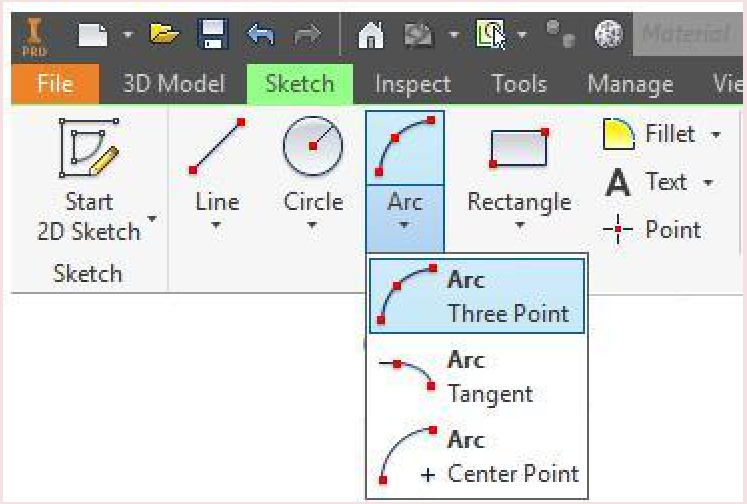
Figure 18-1 Methods of Selecting Objects

Selecting Objects

Up to this point in the book, the only way that has been shown how to select objects was to pick them, one at a time. A quicker and more efficient method of selecting multiple objects is to use either a window or a crossing window. Study Figure 18-1 and start using windows and crossing windows when selecting multiple objects in both Sketch or Model mode.

> Inventor Command: THREE POINT ARC
>
> The THREE POINT ARC command is used to draw an arc by selecting the two endpoints of the arc and a third point anywhere on the circumference of the arc.

Shortcut: **none**



AUTHOR'S COMMENTS: Construct the arc either clockwise or counterclockwise. Point 3 MUST always be on the circumference of the arc, between Point 1 and Point 2. See Figure 18-2.

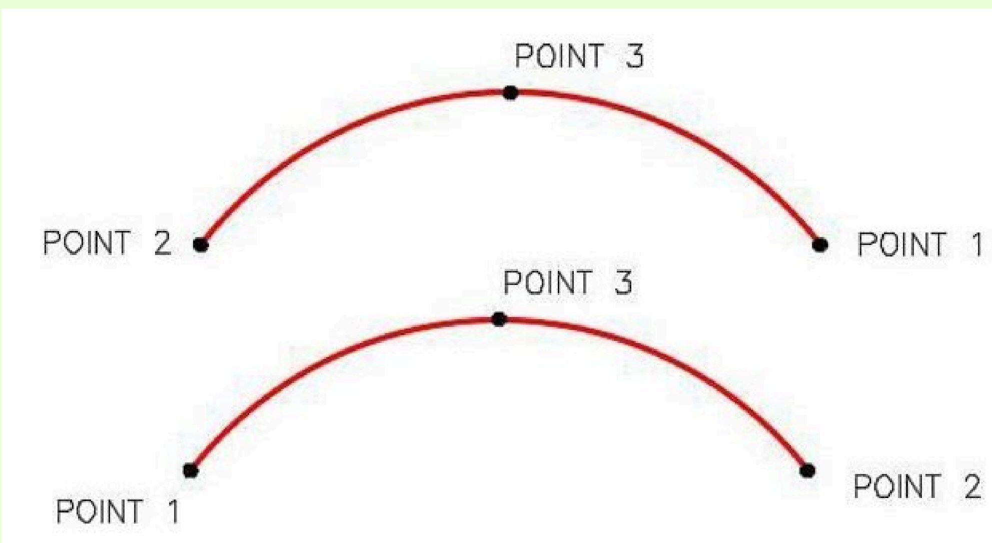
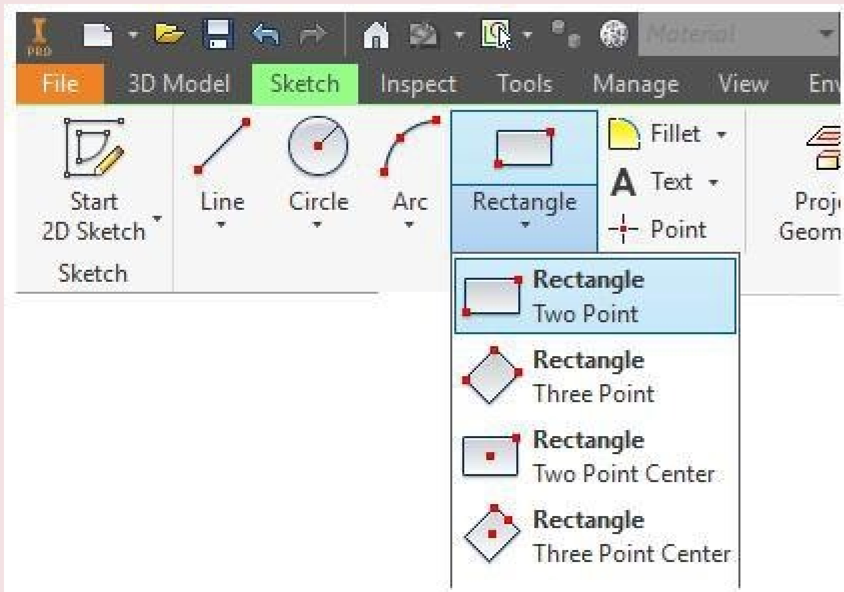


Figure 18-2
Construction Techniques for Arcs

Inventor Command: TWO POINT RECTANGLE

The TWO POINT RECTANGLE command is used to draw a rectangle or a square by selecting two points of its opposite corners.

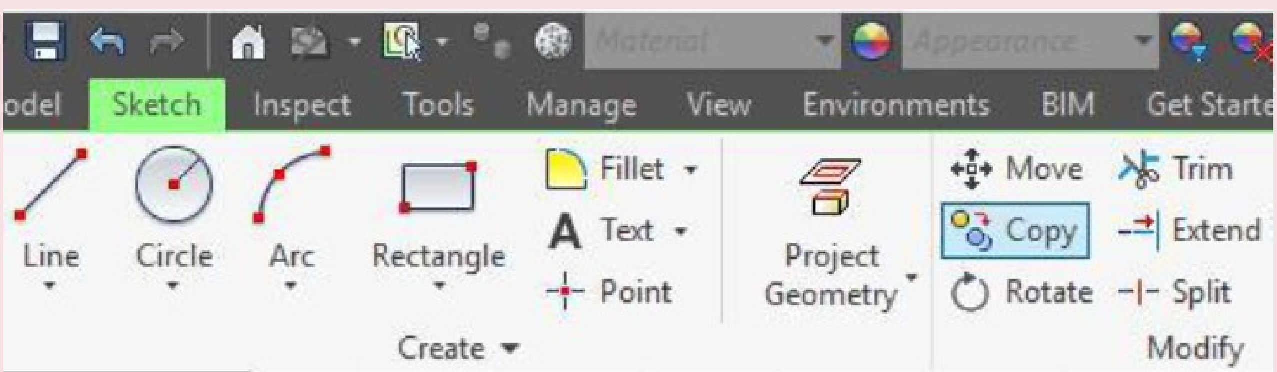
Shortcut: **none**



Inventor Command: COPY

The COPY command is used in Sketch mode to copy 2D geometry from one XY location to another.

Shortcut: **none**



WORK ALONG: Drawing a Solid Model with Arcs and Rectangles

Step 1

Check the current project and if necessary, set it to Inventor Course.

Step 2

Using the NEW command, start a new part file using the template: English-Modules Part (in).ipt.

Step 3

Save the file with the name: Inventor Workalong 18-1. (Figure Step 3A, 3B, 3C, and 3D)

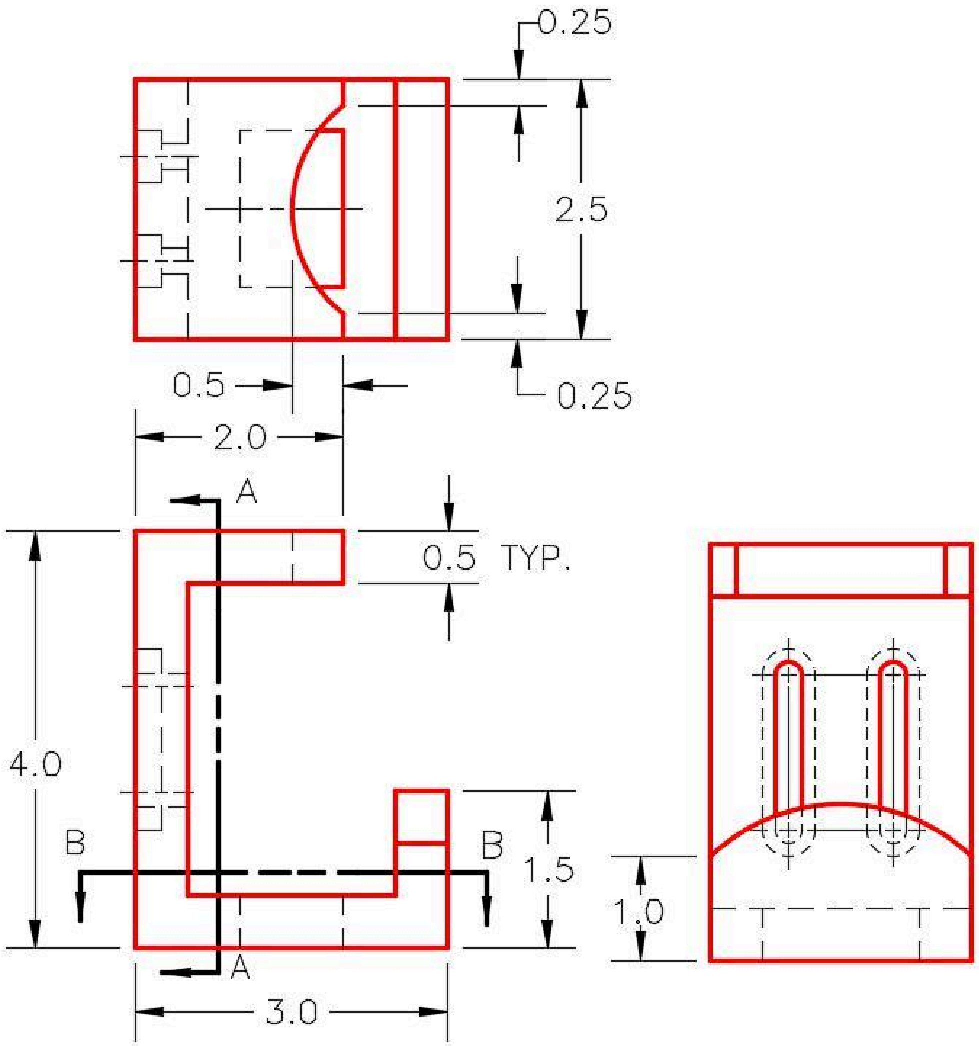


Figure Step 3A
Dimensioned Multiview Drawing [Click to see image full size]

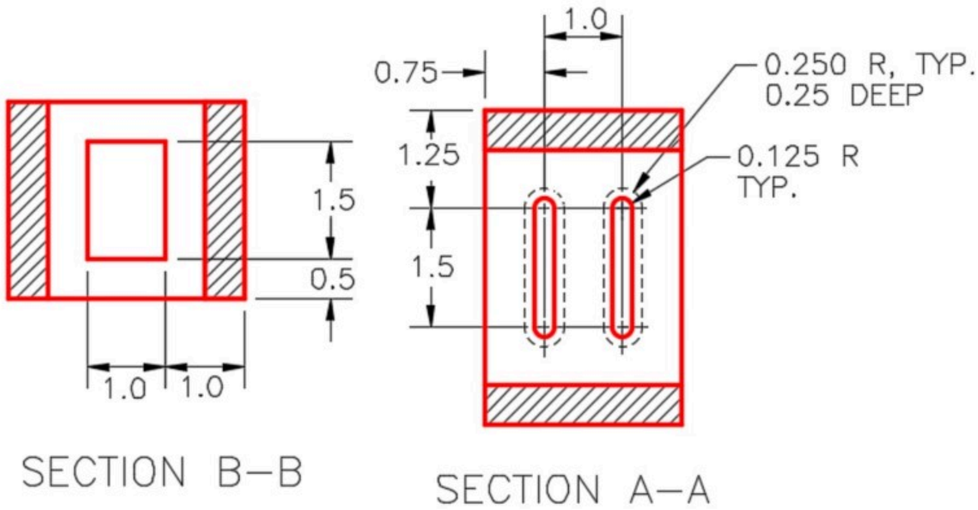


Figure Step 3B
Dimensioned Detail Drawing [Click to see image full size]

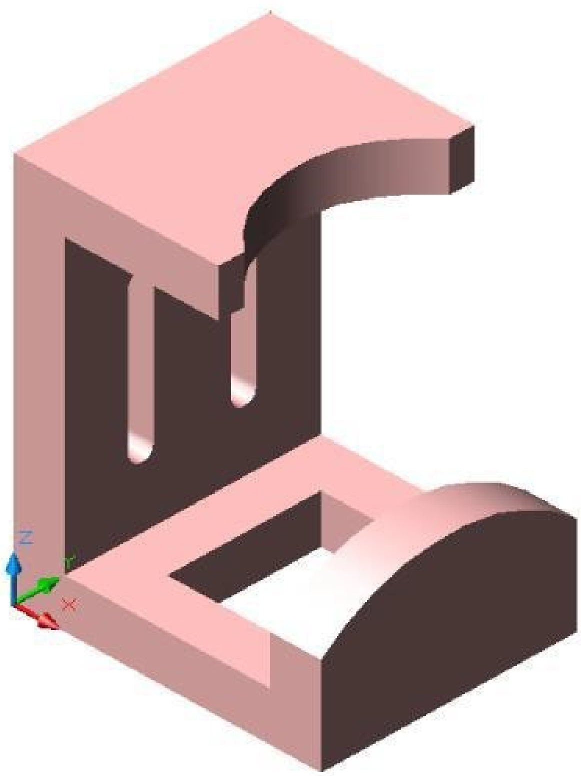


Figure Step 3C
3D Model – Home View [Click to see image full size]

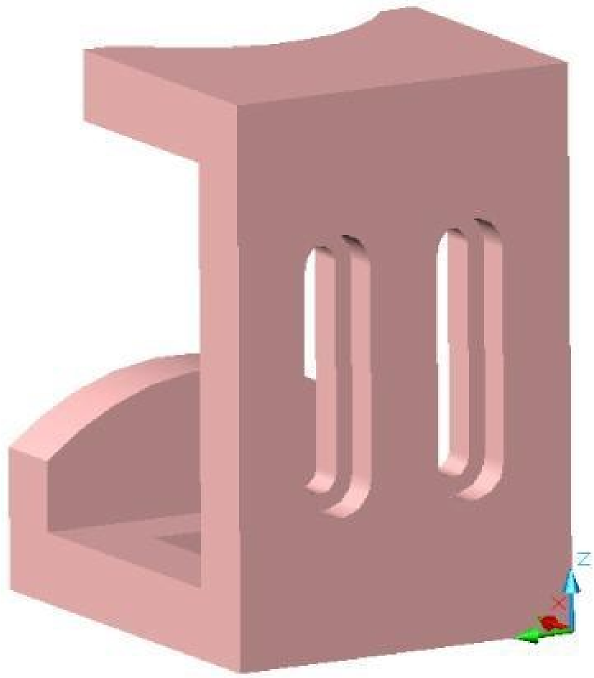


Figure Step 3D
3D Model – Rotated View [Click to see image full size]

Step 4

Start the Base sketch on the Front or XZ plane. Project the Center Point onto the sketch.

Step 5

Draw the four outside lines of the Base sketch. (Figure Step 5)

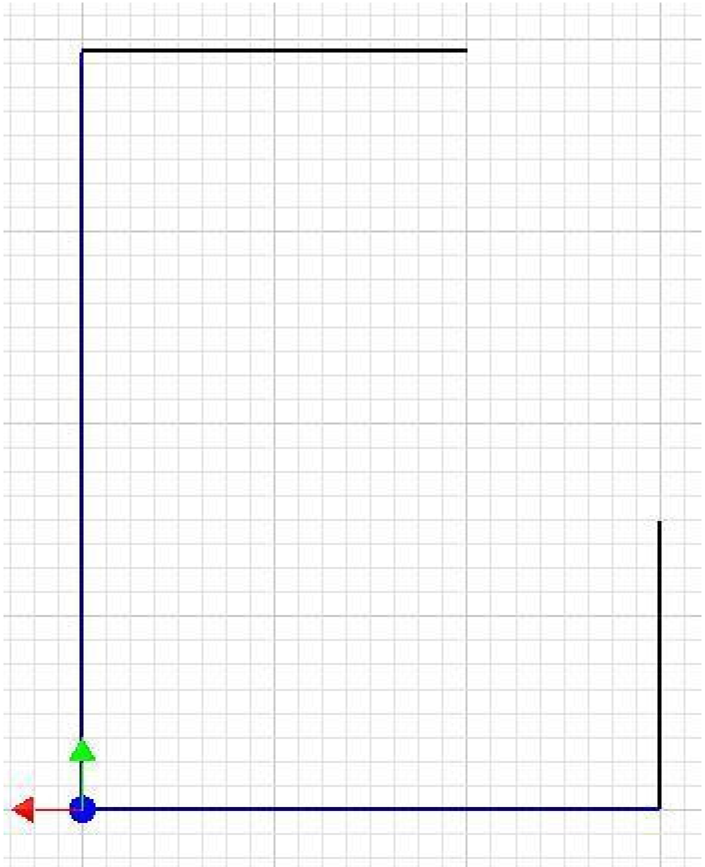


Figure Step 5

Step 6

Offset all four lines as shown in the figure. (Figure Step 6)

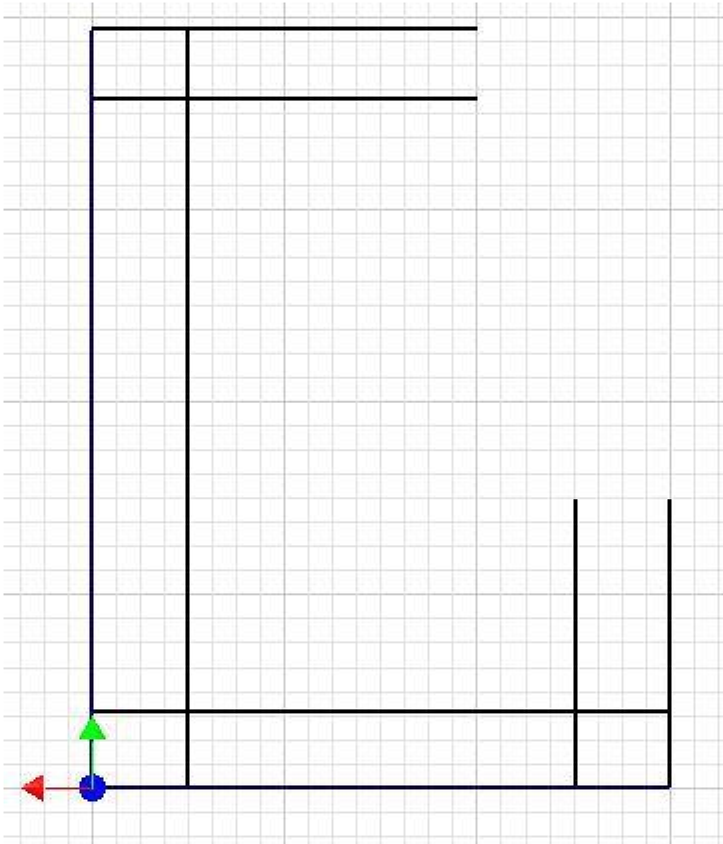


Figure Step 6

Step 7

Trim and dimension the sketch ensuring that it is totally constrained. (Figure Step 7)

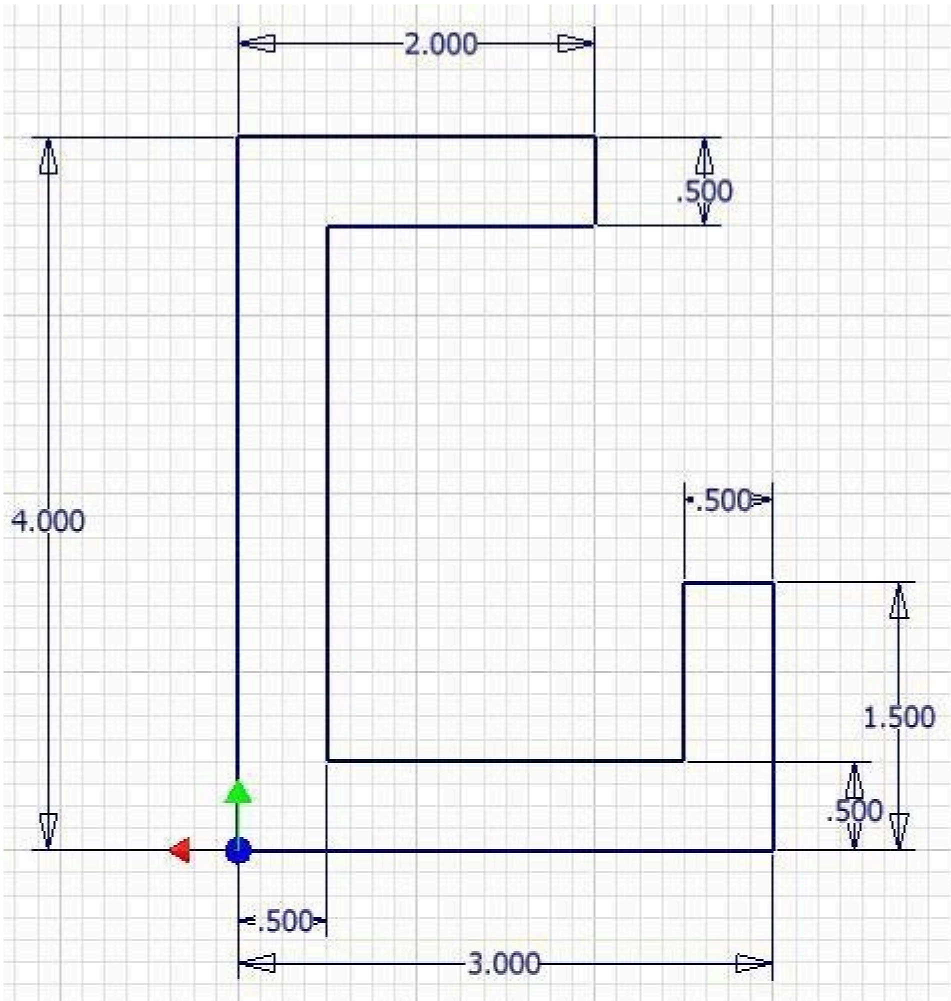


Figure Step 7

Step 8

Press F6 to change the view to the Home view. All of the lines in the sketch should appear purple. Extrude the sketch. (Figure Step 8)

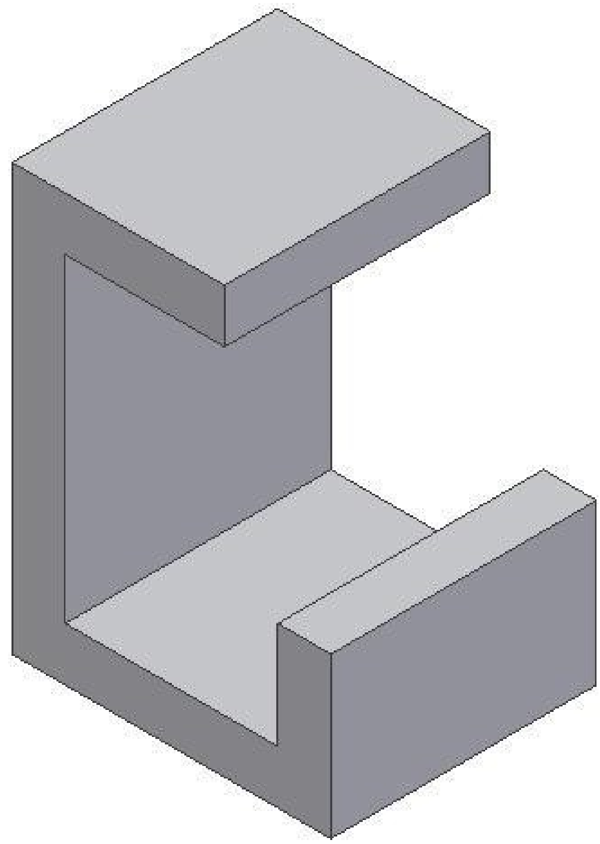


Figure Step 8

Sketching Without the Grid

Up to this point in the Inventor book, the grid display has been enabled in Sketch mode. To the more experienced operator, grid display can be disabled in Sketch mode. On some sketches, it is sometimes easier to draw without the grid getting in the way.

Step 9

Start a new sketch on the right side face as shown in the figure. (Figure Step 9)

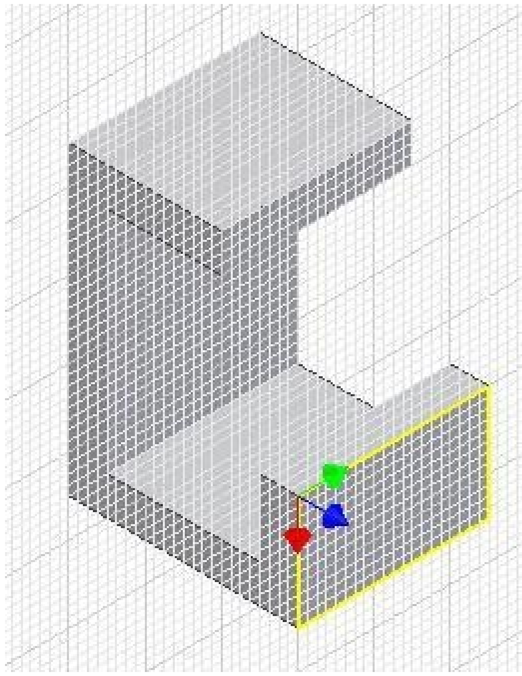


Figure Step 9

Step 10

Using what you already learned, open the Application Options dialogue box.

Step 11

Enable the Sketch tab.

Step 12

In the Display area, disable Grid Lines. (Figure Step 12)

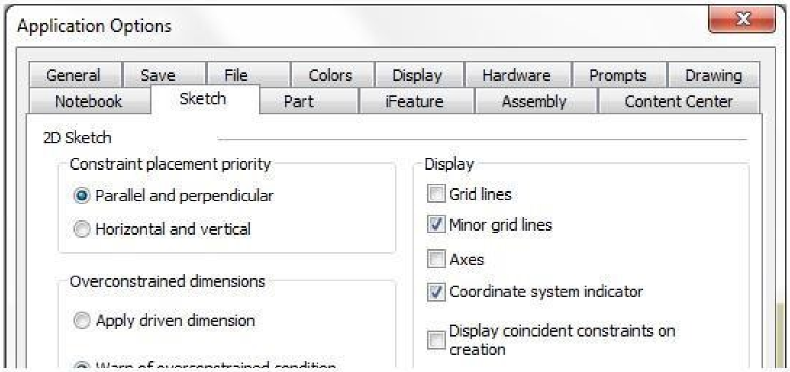


Figure Step 12

> **AUTHOR'S COMMENTS:** Disabling the grid display in a sketch can help when drawing some sketches.

Step 13

Draw a construction line 0.5 inches below the top edge and dimension it. Ensure that it is fully constrained. (Figure Step 13)

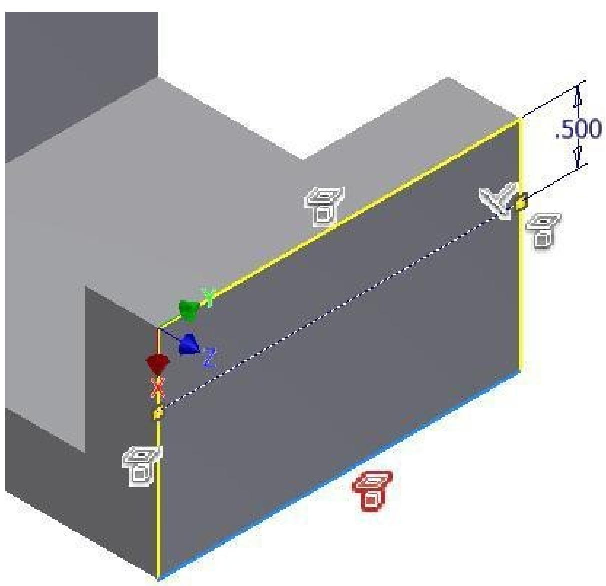


Figure Step 13 [Click to see image full size]

AUTHOR'S COMMENTS: Do not offset an edge. Draw the line, using the LINE command, constrain it perpendicular to one of the edges.

Step 14

Click the THREE POINT ARC command. Snap the first point to one end of the construction line, snap the second point to the other end of the construction line. For the third point, snap it to the midpoint of the top edge. (Figure Step 14A, 14B, and 14C)

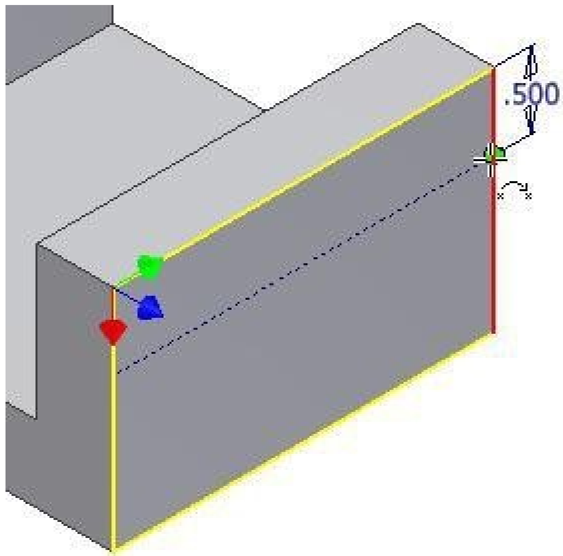


Figure Step 14A

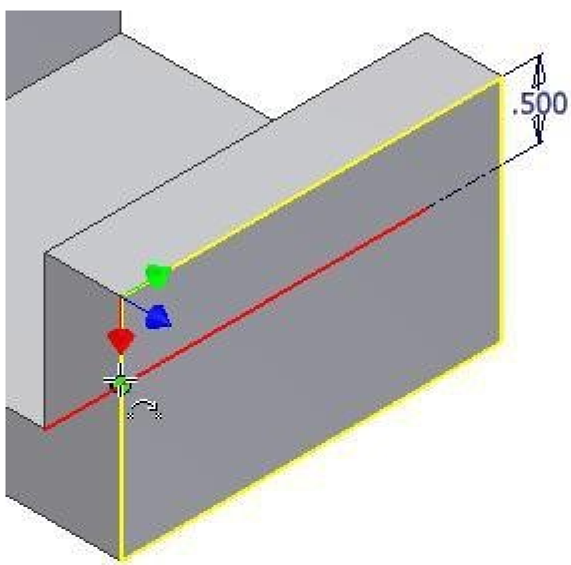


Figure Step 14B

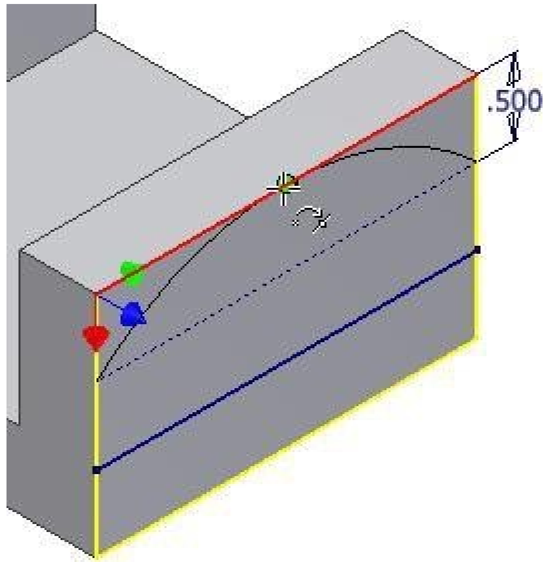


Figure Step 14C

Step 15

Using the CREATE CONSTRAIN command, apply the Tangent constraint to the arc and the top edge. This should fully constrain the sketch. (Figure Step 15)

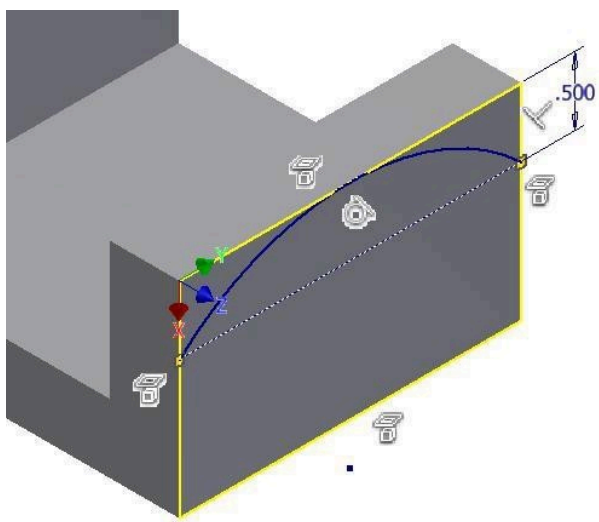
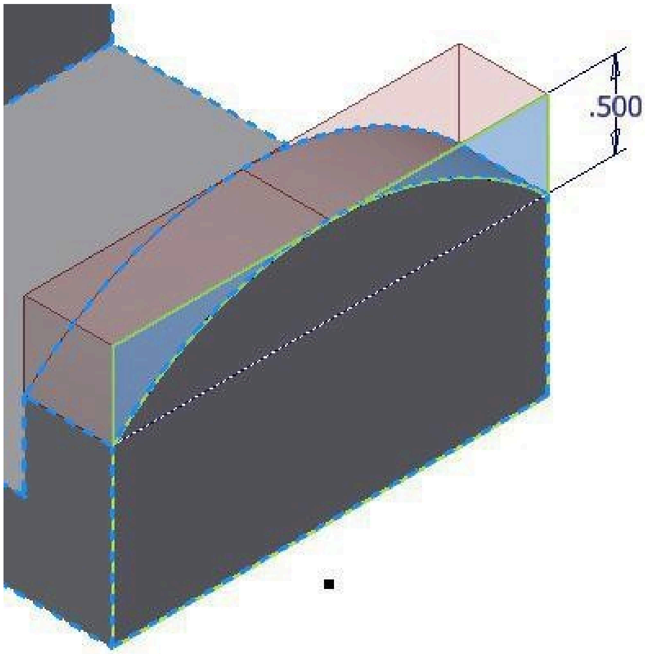


Figure Step 15 [Click to see image full size]

Step 16

Extrude the sketch. (Figure Step 16A and 16B)



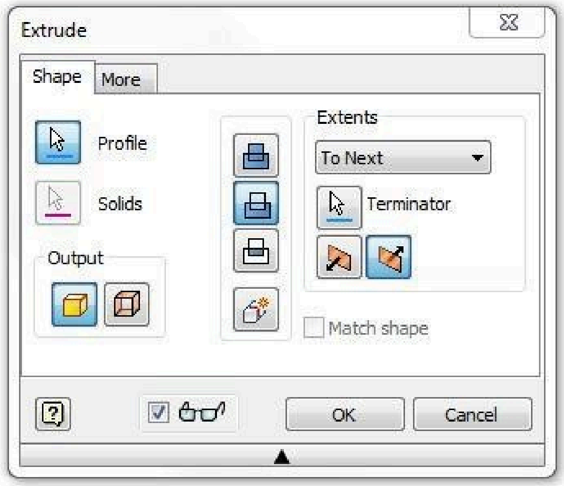


Figure Step 16A [Click to see image full size]

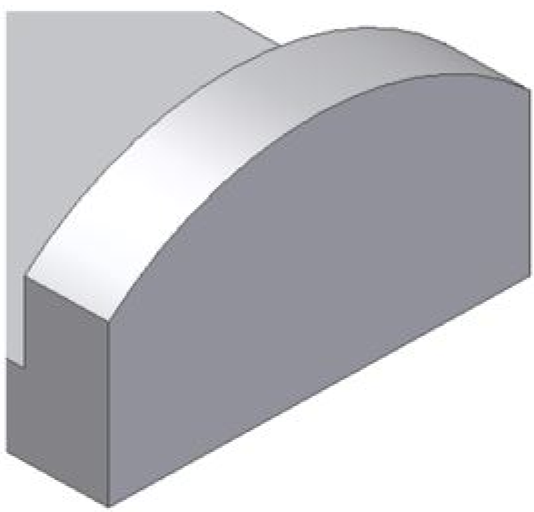


Figure Step 16B

Step 17

Using what you just learned, start a new sketch on the top of the model. On it, draw three construction lines and a three point arc. Ensure that the sketch is fully constrained. Extrude the sketch. (Figure Step 17A and 17B)

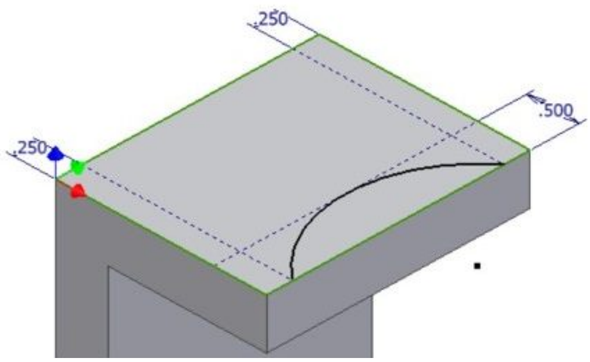


Figure 17A [Click to see image full size]

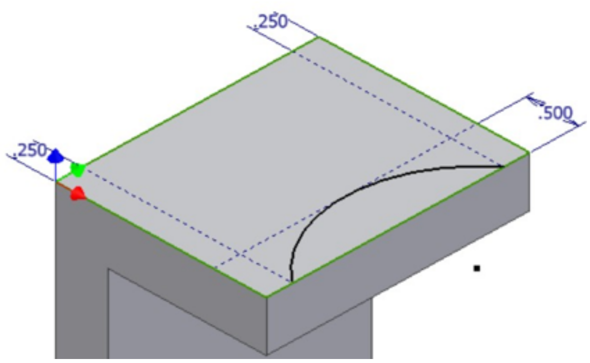


Figure 17B

Step 18

Start a new sketch on the lower plane. (Figure Step 18)

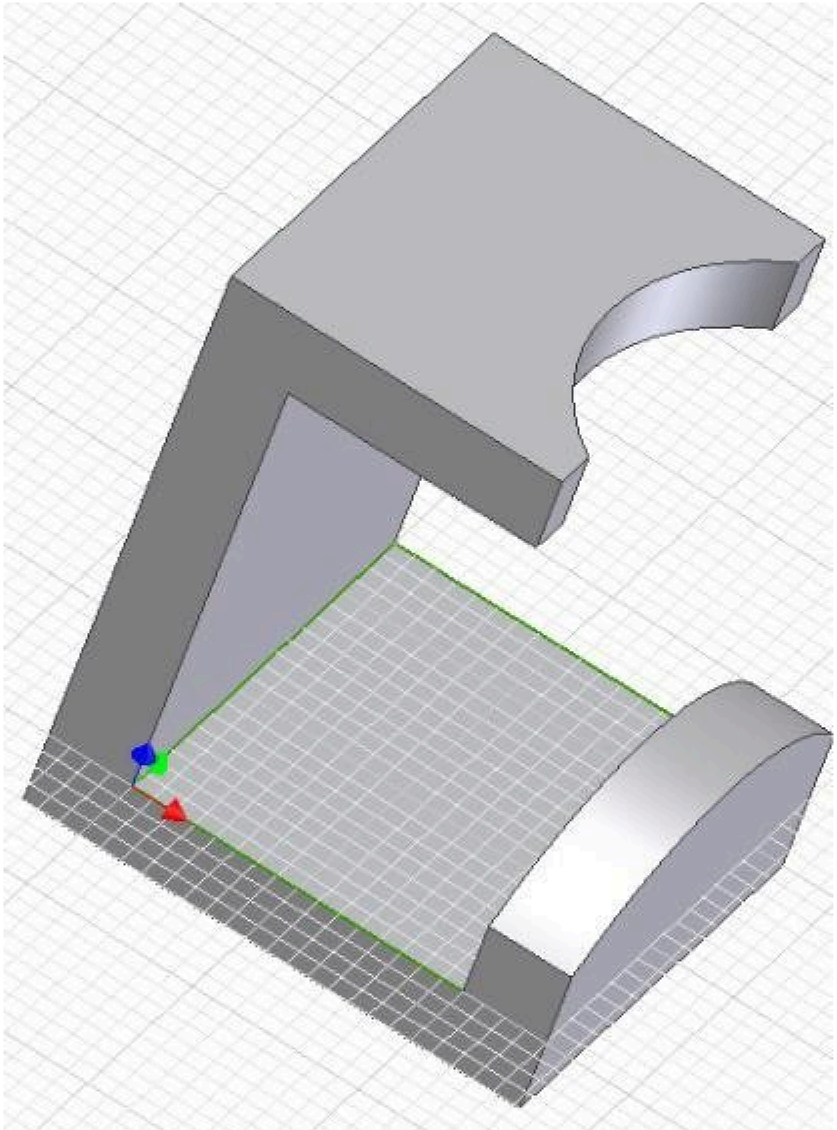


Figure Step 18

AUTHOR'S COMMENTS: I have shown the new sketch with the grid displayed only to help you visualize what plane the sketch is on.

Step 19

On the sketch, using the TWO POINT RECTANGLE command, draw a rectangle by selecting two opposite corners. Guess at the location and size. (Figure Step 19)

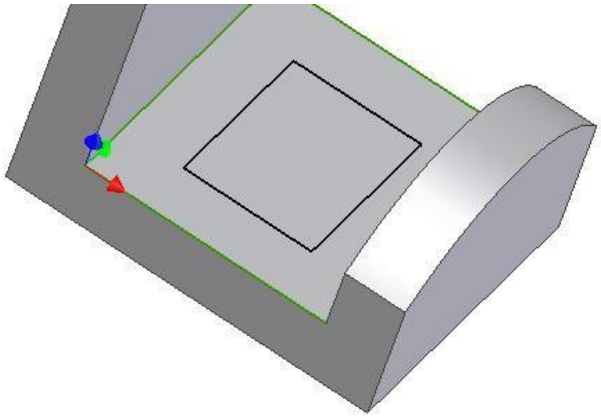


Figure Step 19 [Click to see image full size]

Step 20

Insert four dimensions, one from each edge. Check to ensure that the sketch is fully constrained and extrude it. (Figure Step 20A and 20B)

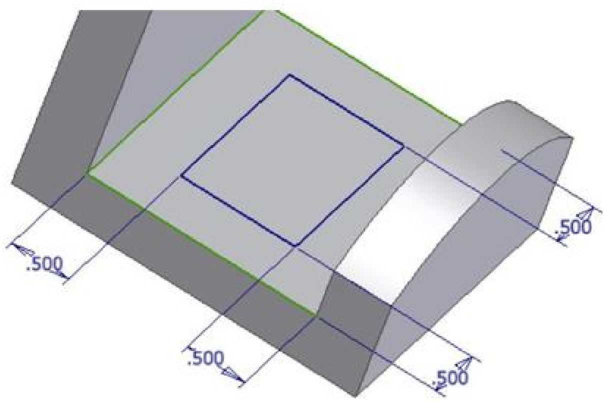


Figure Step 20A [Click to see image full size]

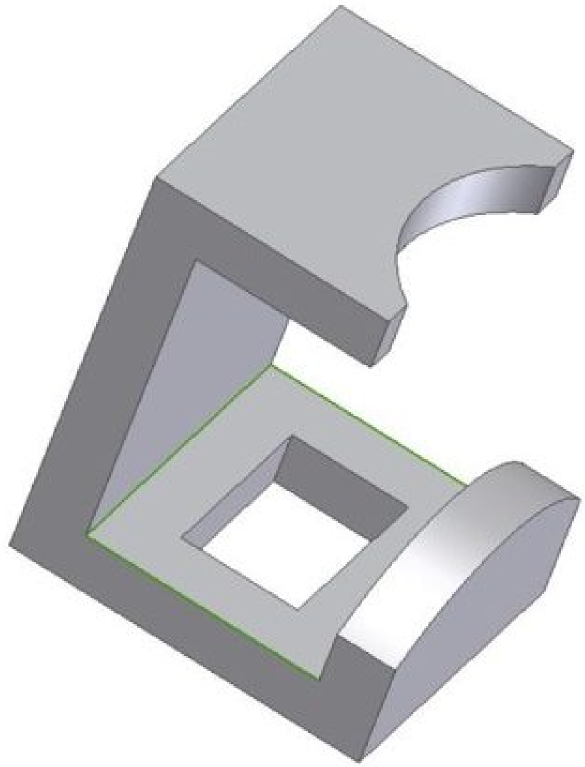


Figure Step 20B

Step 21

Start a new sketch on the back plane and place a construction line as shown in the figure. Do not offset the edge. Ensure that you constrain the line perpendicular to the bottom or top edge. You can guess at the start point and the length of the line. (Figure Step 21)

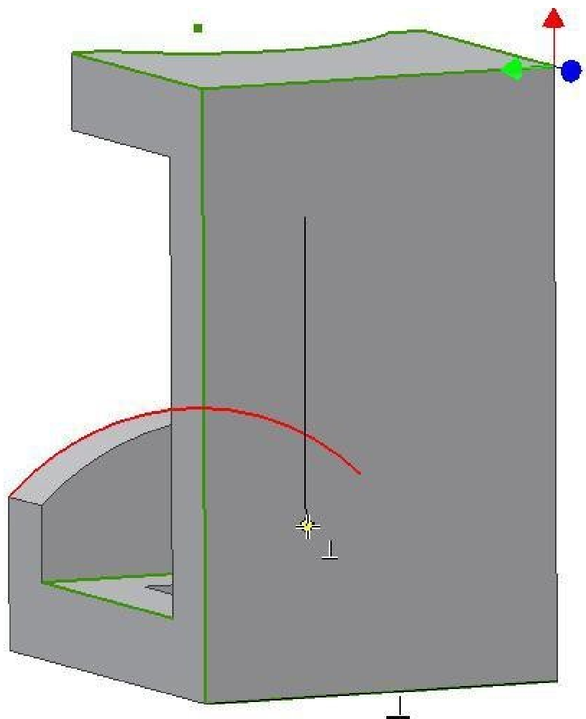


Figure Step 21 [Click to see image full size]

Step 22

Change the property of the line to a construction line. Insert one circle at the each end of the line. Snap to the end of the line to locate the centre of the circles. (Figure Step 22)

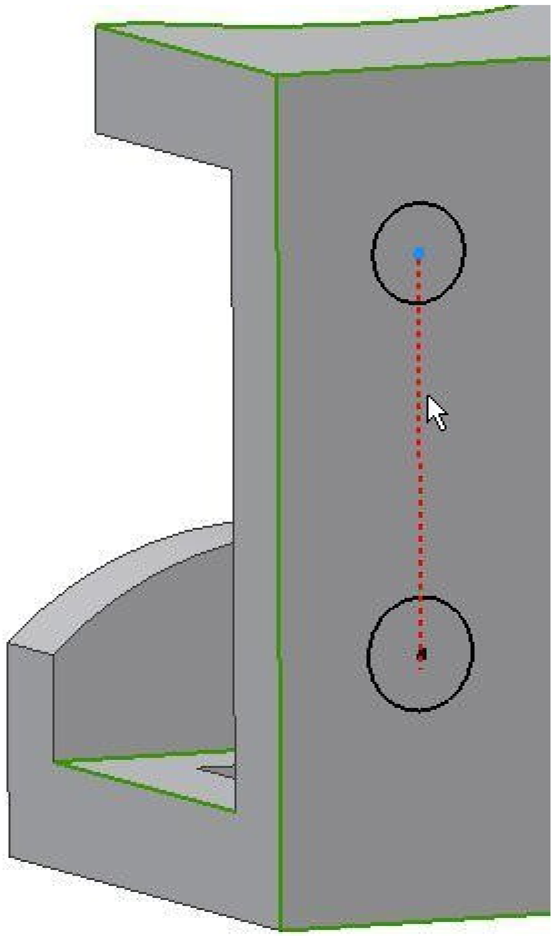


Figure Step 22

Step 23

Using what you learned already, dimension one of the circles and then create an Equal constraint for the other one. Draw lines on either side of the circle. Ensure that you snap the end of the lines onto the circles. Constrain the lines to the circle with the Tangent constraint. Trim the circles. (Figure Step 23A, 23B, and 23C)

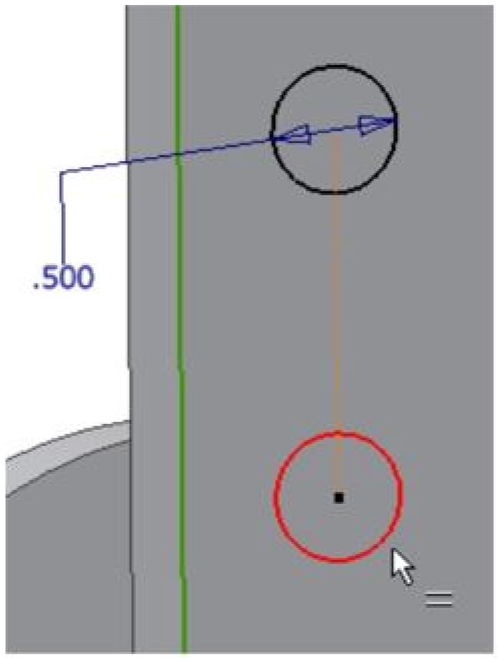


Figure Step 23A

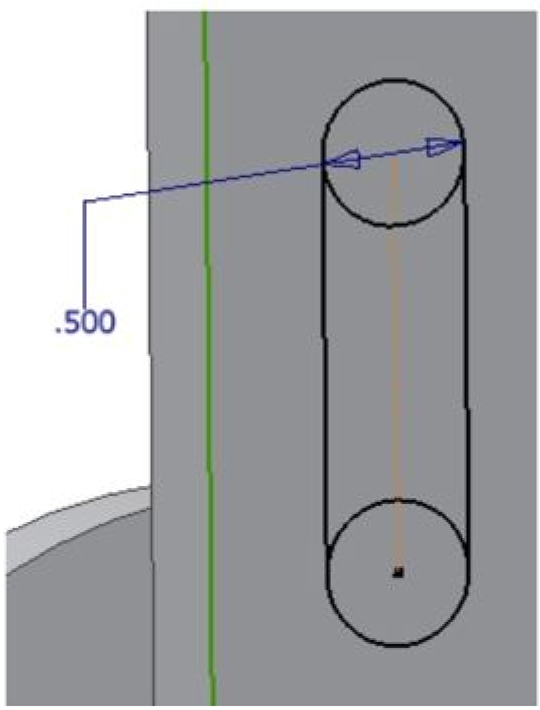


Figure Step 23B

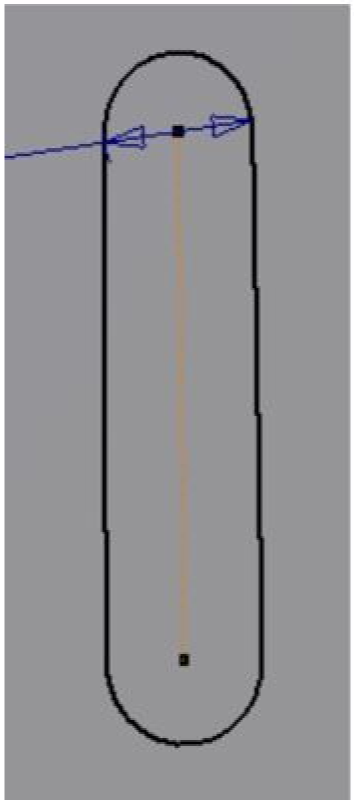


Figure Step 23C

Step 24

Draw a construction line from the end of the construction line at the top circle, constraining it perpendicular to one of the vertical lines. Draw it approximately 1 inch long. (Figure Step 24)

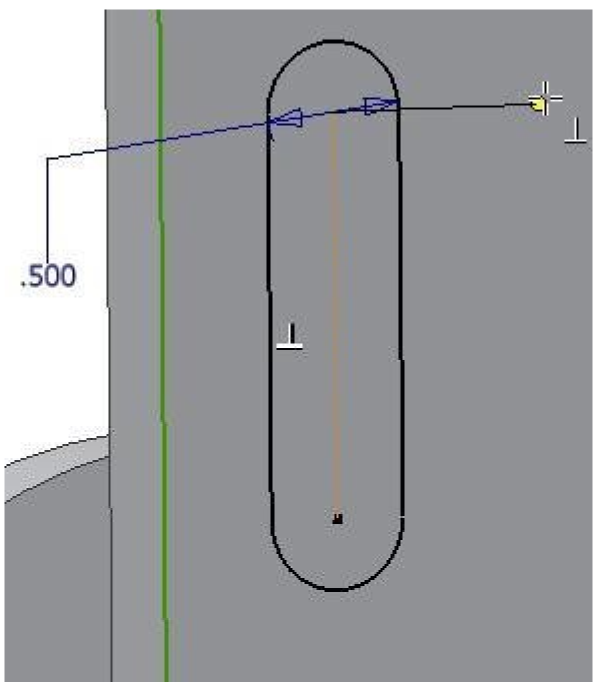


Figure Step 24

Step 25

Click the COPY command and select the two lines and the arcs as shown in the figure. Ensure that the Select icon is enabled. (Figure Step 25)

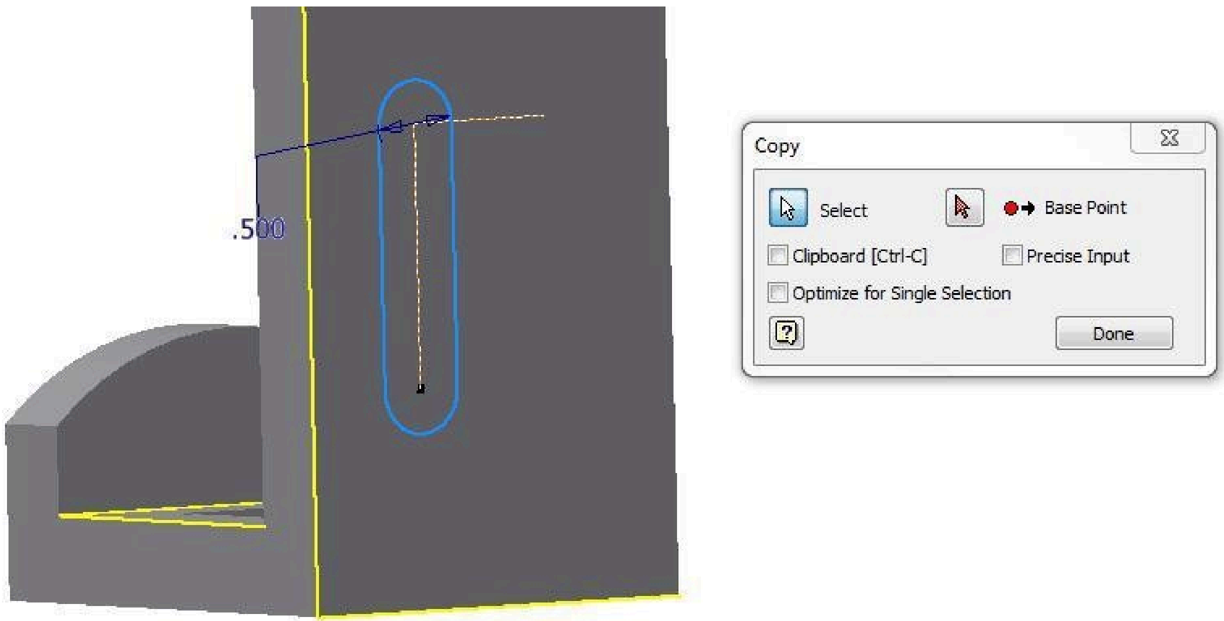


Figure Step 25 [Click to see image full size]

Step 26

Enable the From icon and select the end of the construction line for the From location. (Figure Step 26)

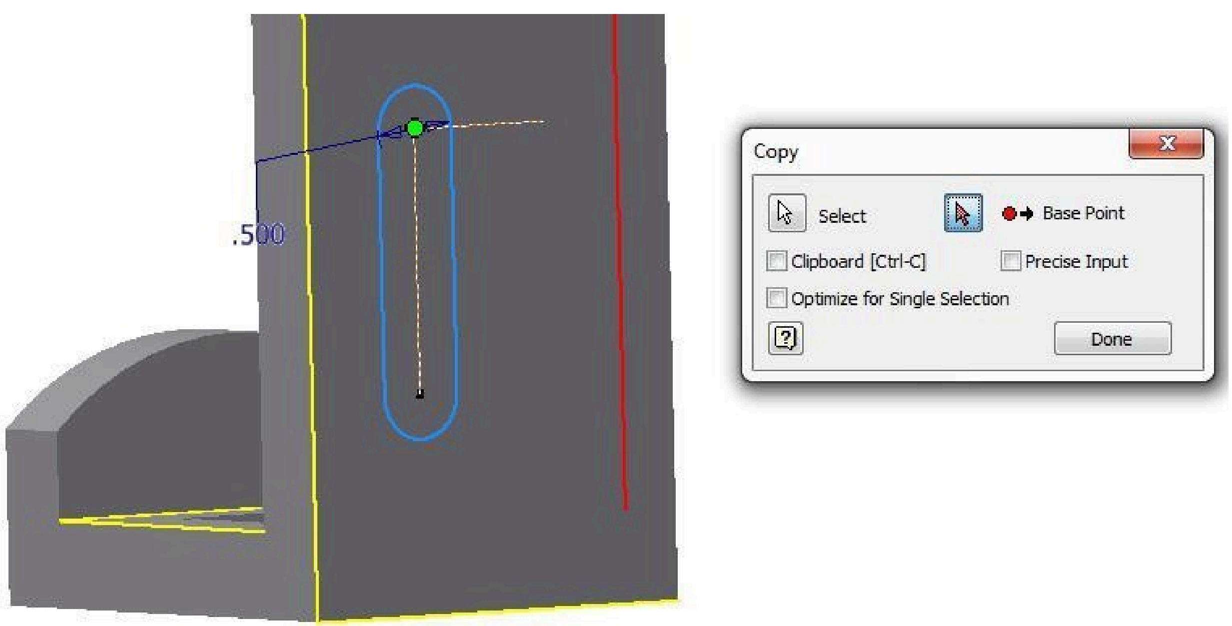
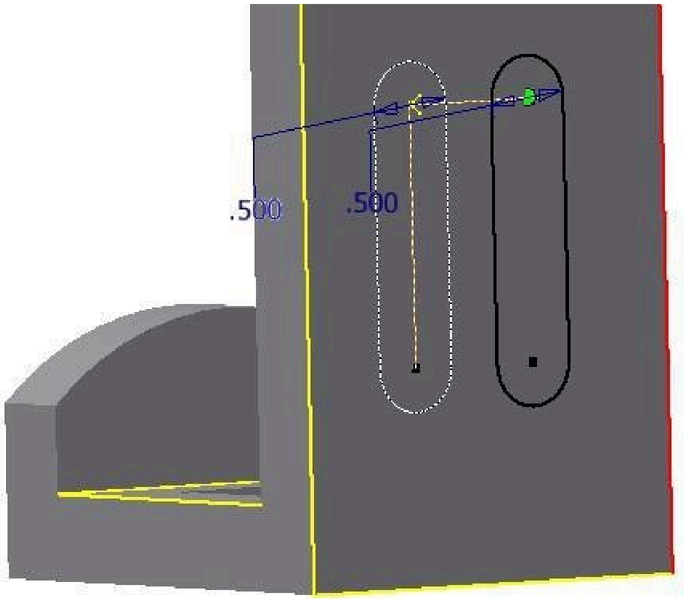


Figure Step 26 [Click to see image full size]

Step 27

Enable the To icon and select the other end of the construction line for the To location. (Figure Step 27)



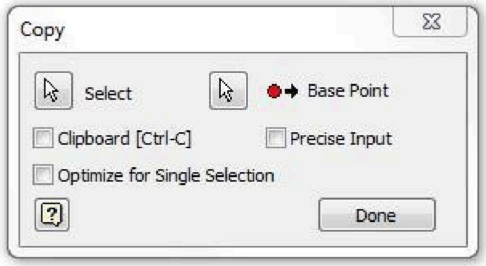


Figure Step 27

Step 28

If you get the warning message, click Yes.

Step 29

Dimension the sketch until it is fully constrained. (Figure Step 29)

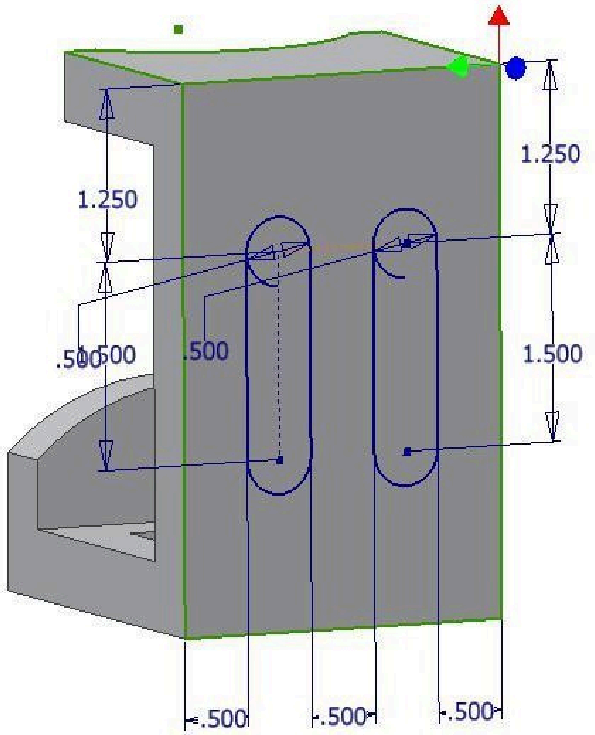


Figure Step 29 [Click to see image full size]

Step 30

Return to Model mode and extrude the slots. Extrude them 0.25 inches deep. (Figure Step 30)

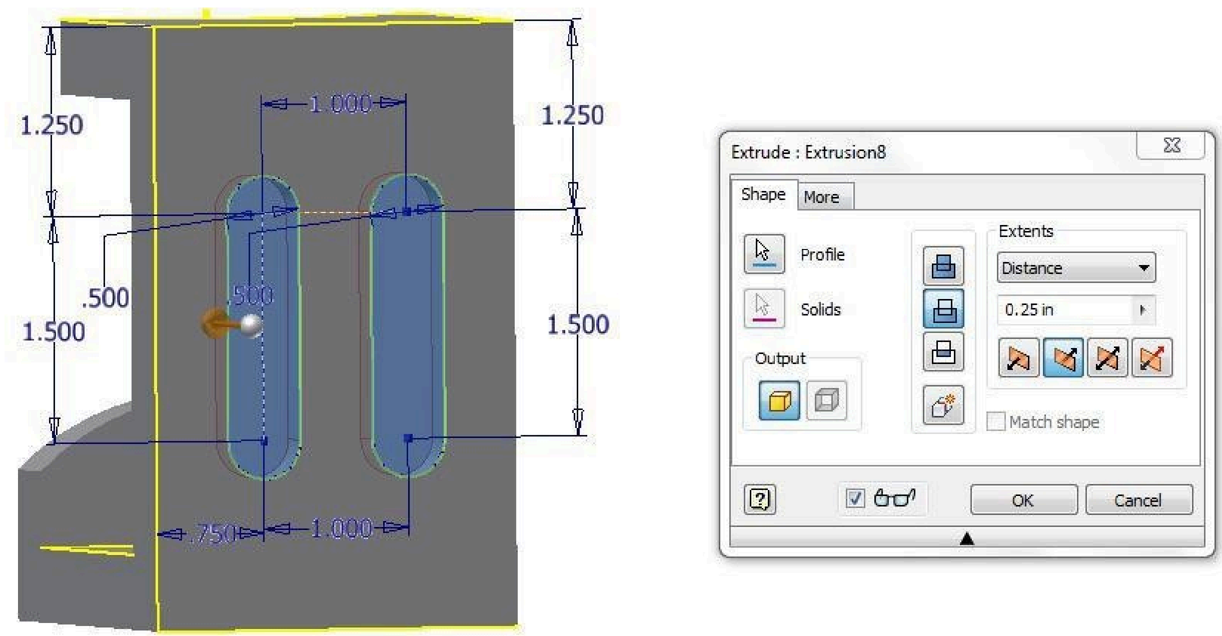


Figure Step 30 [Click to see image full size]

Step 31

Start a new sketch on the bottom of the extruded slots. Enter the OFFSET command and right-click one of the arcs. In the Right-click menu, enable Loop Select. Select the other arc and the lines to complete the loop. (Figure Step 31A and 31B)

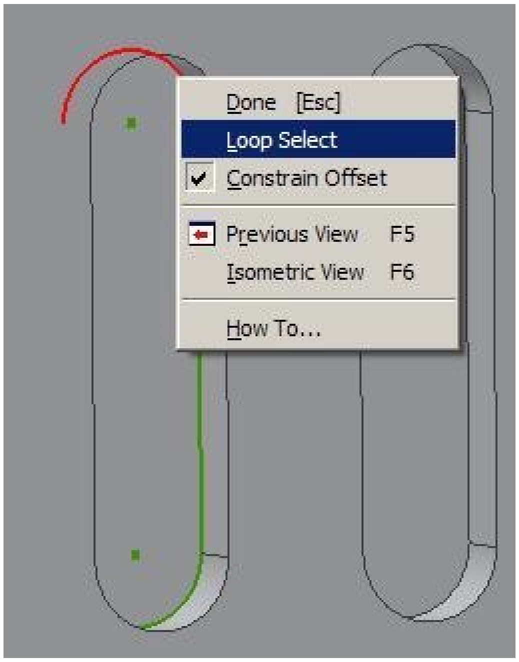


Figure Step 31A

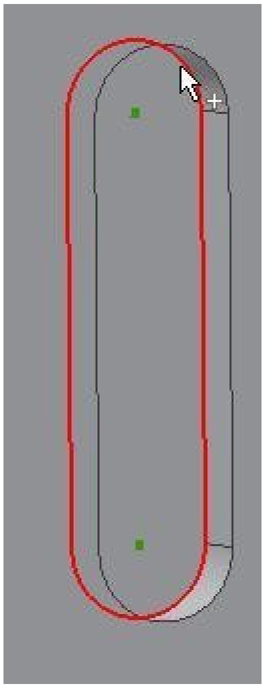


Figure Step 31B

Step 32

Offset the slot towards the inside. Guess at the offset distance. (Figure Step 32)

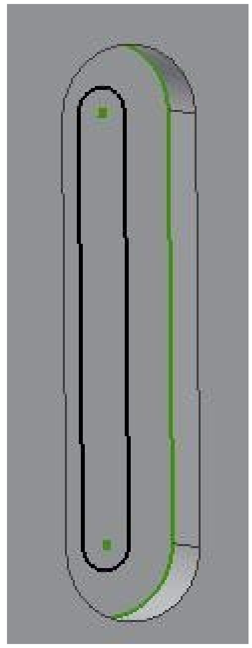


Figure Step 32

Step 33

Dimension the offset. (Figure Step 33)

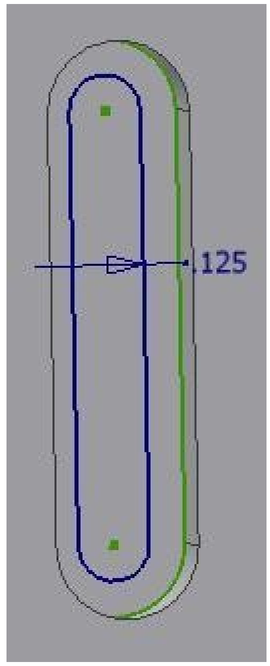


Figure Step 33

Step 34

Extrude the slot To Next. (Figure Step 34)

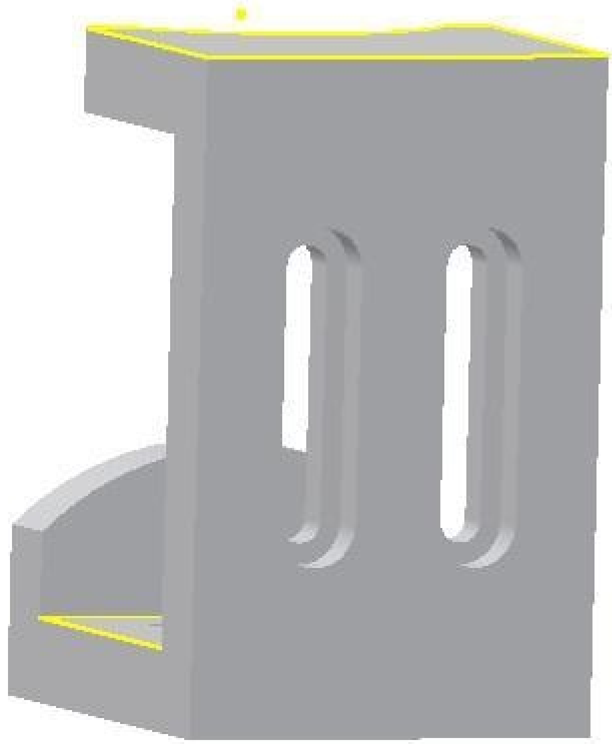


Figure Step 34

Step 35

Redo Step 31 to 34 on the other slot to complete the solid model.

Step 36

Set the colour to: Dark Red. (Figure Step 36)

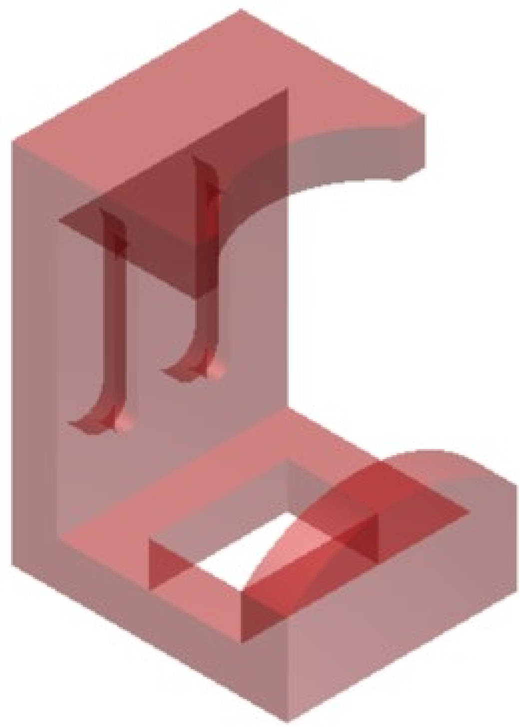


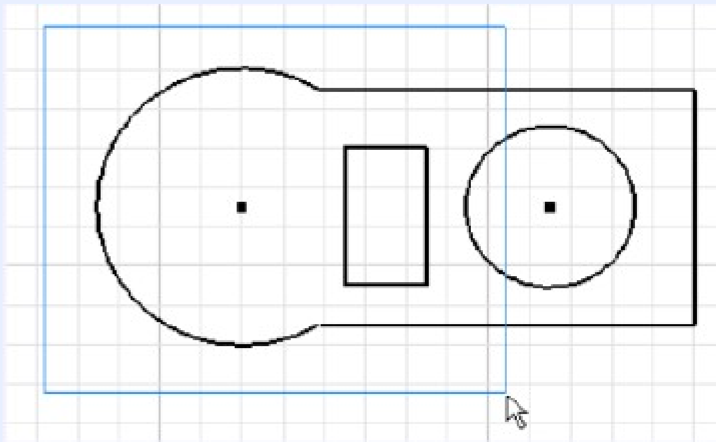
Figure Step 36

Step 37

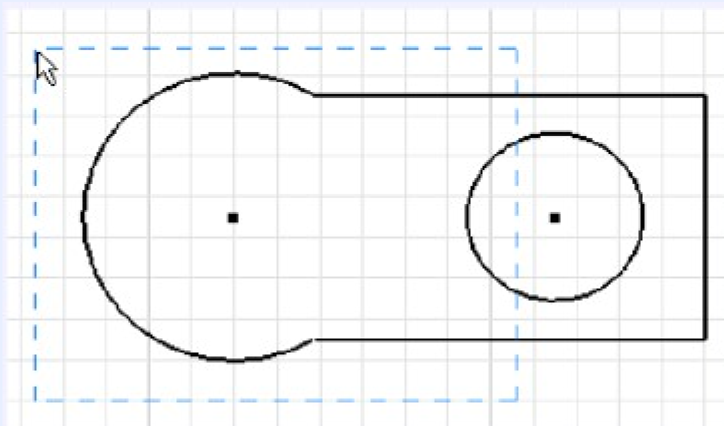
Save and close the file.

MUST KNOW: A window selects all of the objects that are totally inside of it defined by two user picks. A window always appears as a solid line and is created by picking the first point on the left and moving right to pick the second point.A crossing window selects all of the objects that are totally inside and the ones that cross it defined by two user picks. A crossing Window always appear as dashed lines and is created by picking the first point on the right and moving left to select the second point.

To select a window or crossing window, select the first point by moving the cursor to the desired location and press the left mouse button down. While holding it down, move to the cursor to second desired location and release the mouse button.



A Window



A Crossing Window

Key Principles

Key Principles in Module 18

1. A window appears as a solid rectangle selecting only the objects totally inside the window. A crossing window appears as a dashed rectangle selecting all of the objects that it crosses and the ones that are totally inside the crossing window.

Lab Exercise 18-1

Time allowed: 60 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab 18-1	Inventor Course	Millimeters	Metric-Modules Part (mm).ipt	Zinc	N/A

Step 1

Project the Center Point onto the base plane.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude or revolve them to produce the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to maintain the objects shape and size. (Figure Step 2A, 2B, 2C, and 2D)

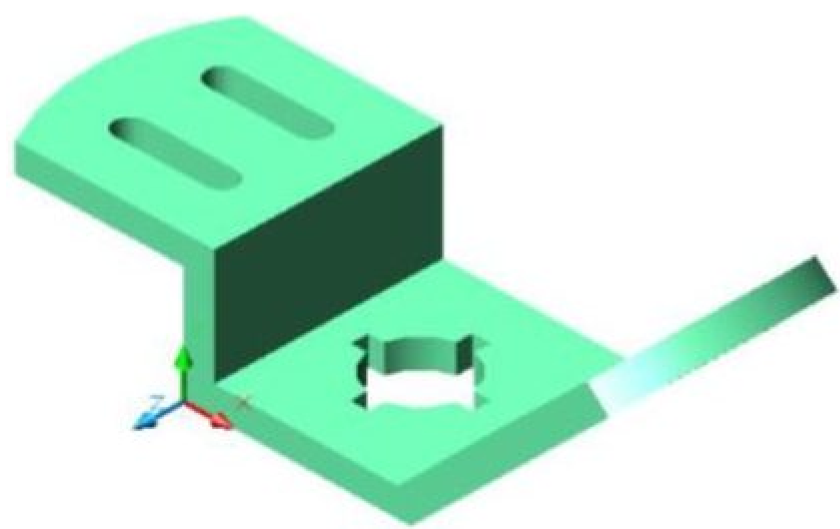


Figure Step 2A
3D Model – Home View

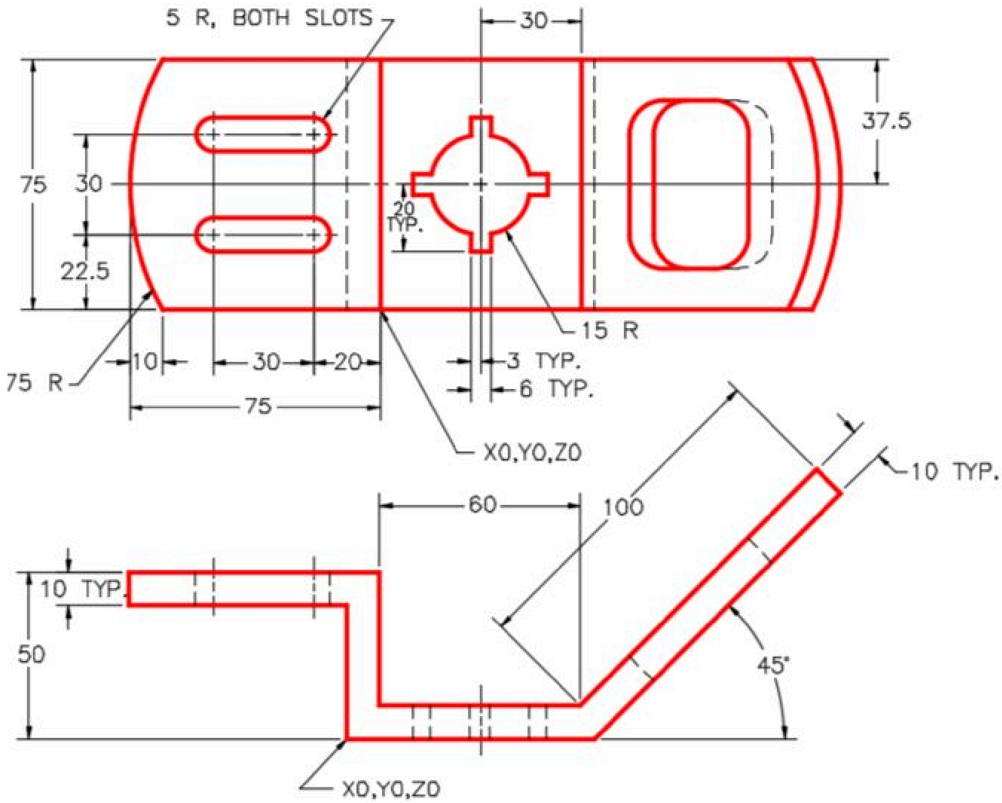


Figure Step 2B
Dimensioned Multiview Drawing [Click to see image full size]

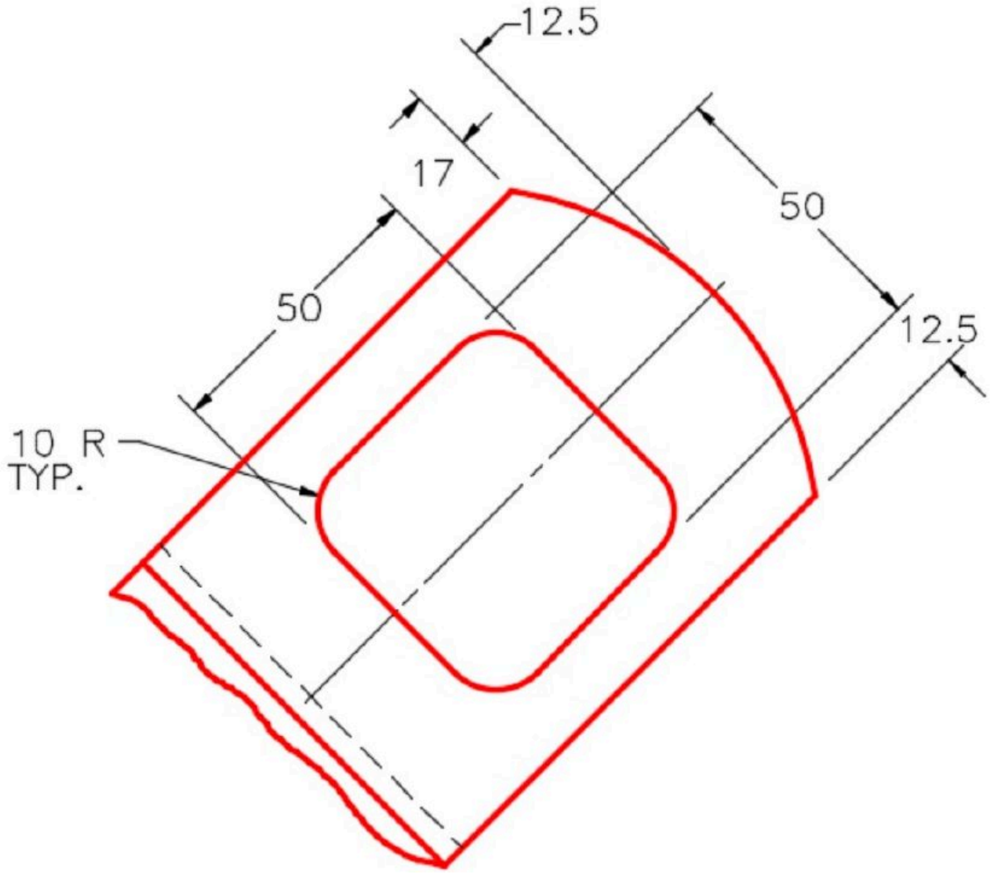


Figure Step 2C
Dimensioned Auxiliary View [Click to see image full size]

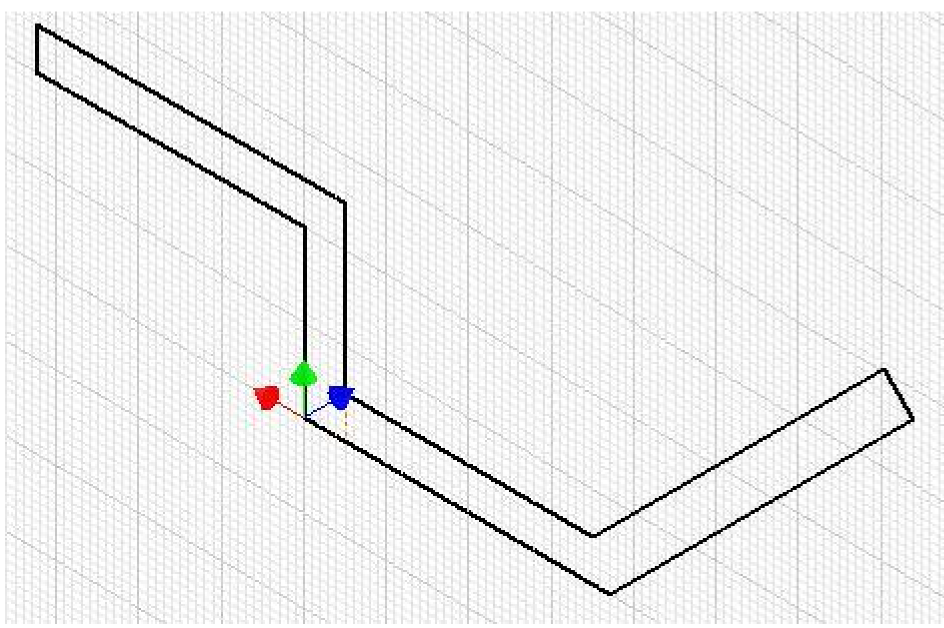


Figure Step 2D
Suggested Base Sketch –
Front (XZ) Plane

Step 3

Apply the colour shown above. (Figure Step 3A and 3B)

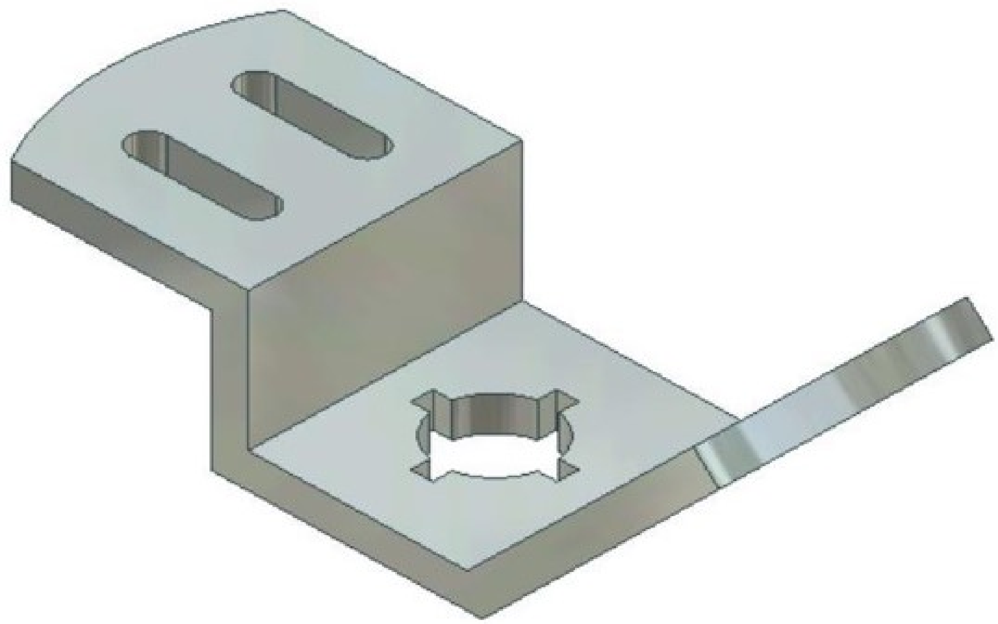


Figure Step 3A
Solid Model – Home View

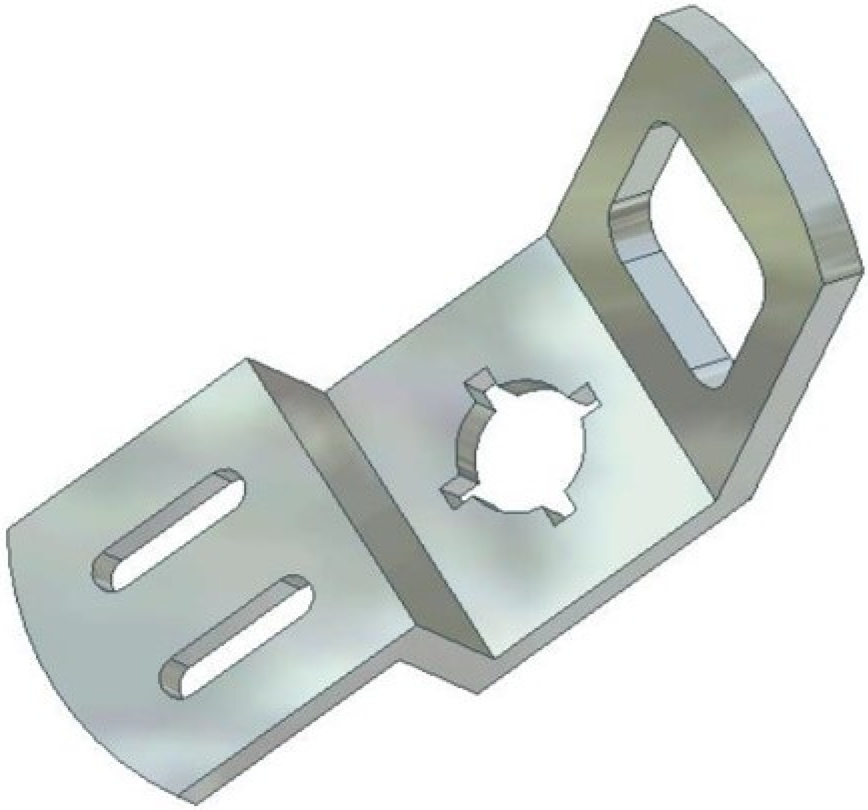


Figure Step 3B
Solid Model – Rotated View

Lab Exercise 18-2

Time allowed: 60 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab 18-2	Inventor Course	Inches	English-Modules Part (in).ipt	Galvanized (texture)	N/A

Step 1

Project the Center Point onto the base plane.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude or revolve them to produce the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to maintain the objects shape and size. (Figure Step 2A and 2B)

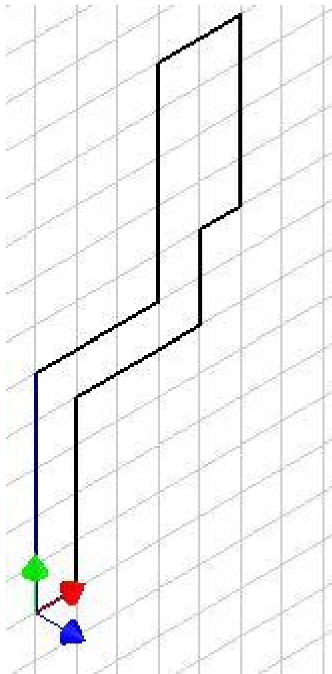


Figure Step 2A Suggested Base Sketch – Right Side (YZ Plane)

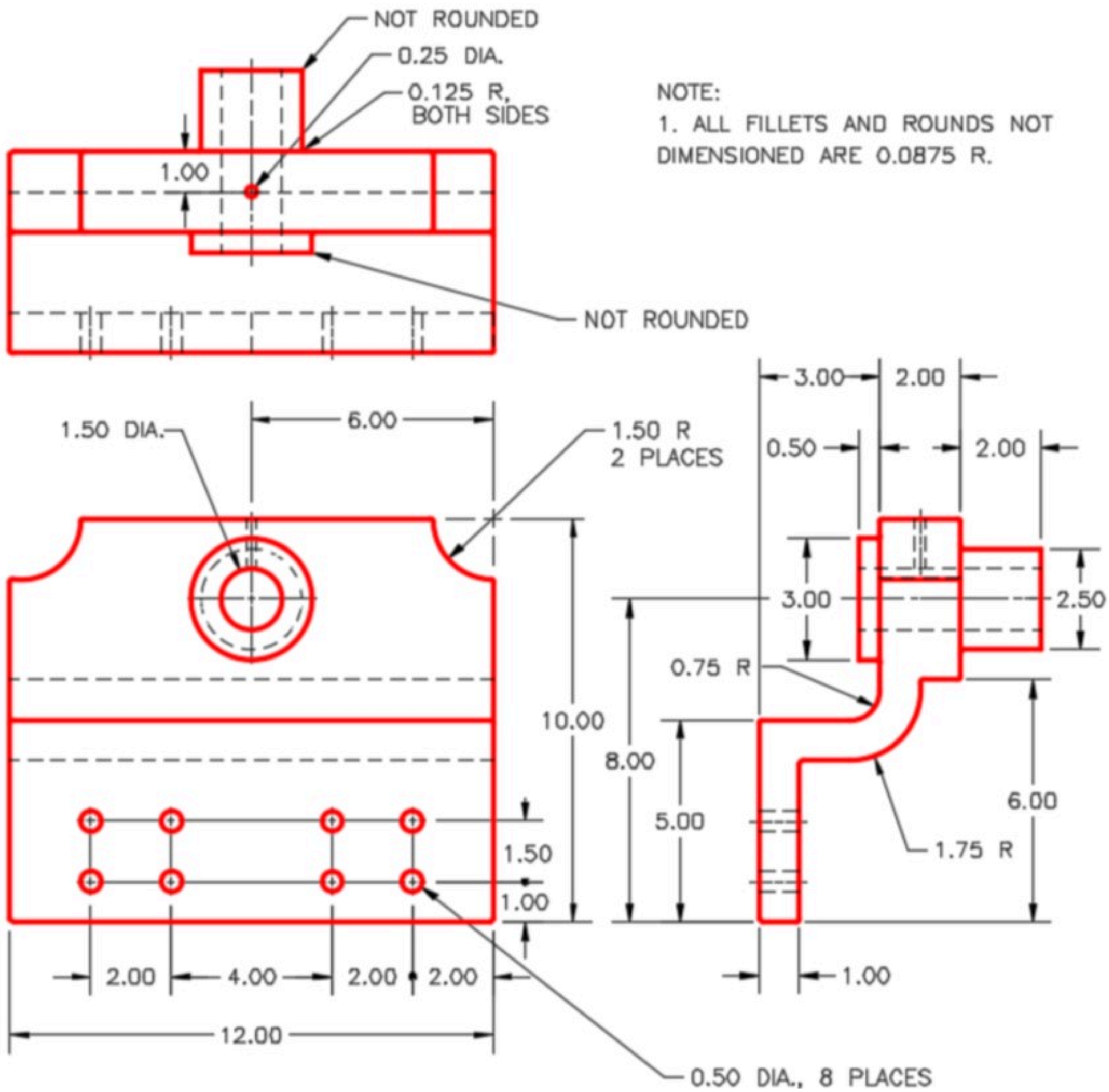
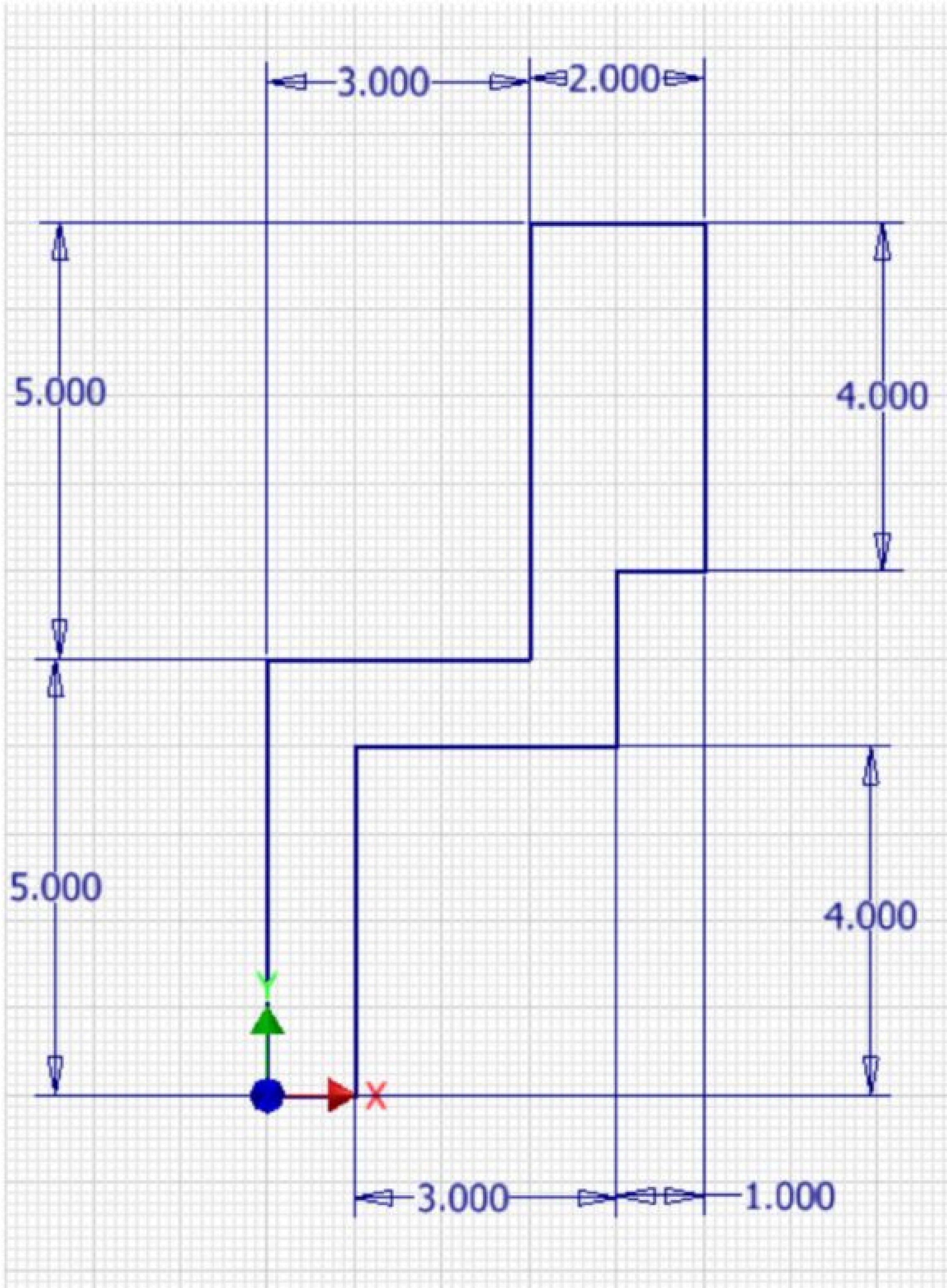
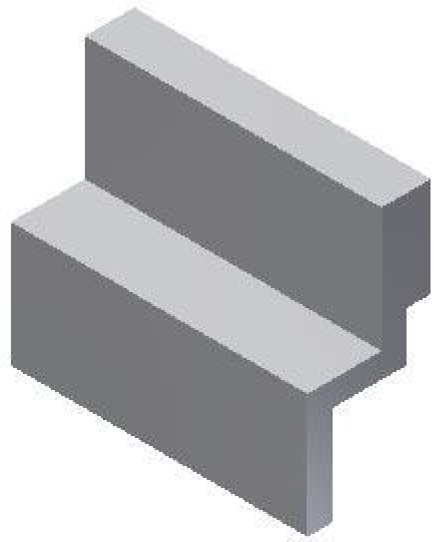


Figure Step 2B
Dimensioned Multiview Drawing



Author`s Base Sketch



Author’s Base Model

Step 3

Apply the colour shown above. (Figure Step 3A and 3B)

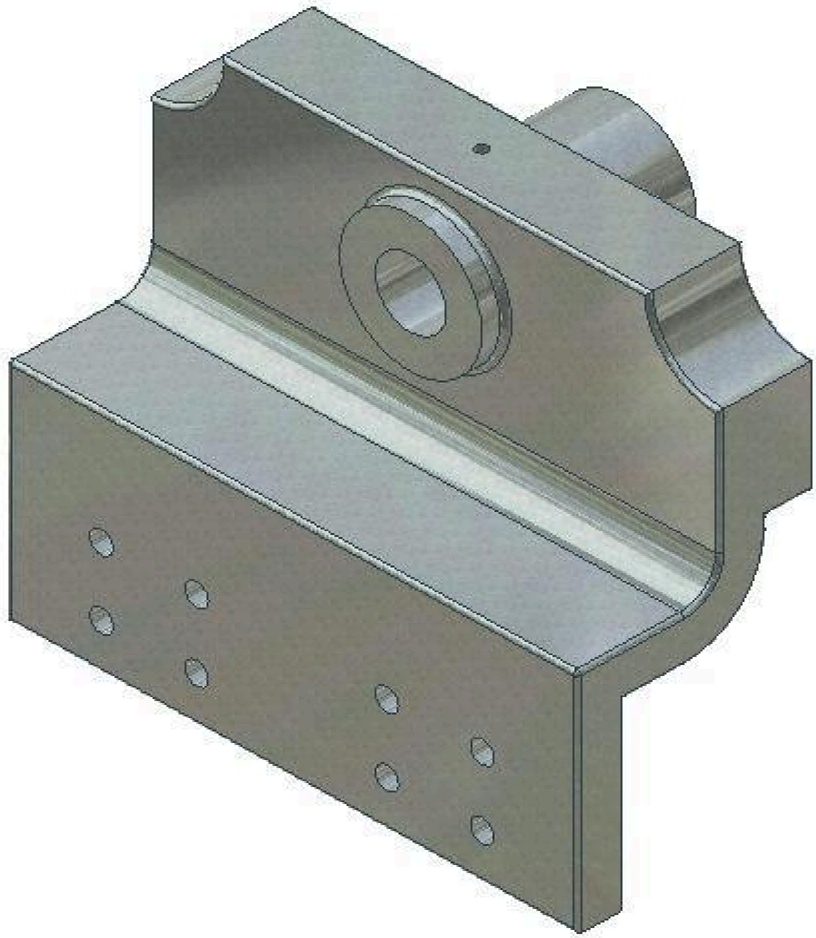


Figure Step 3A
Solid Model – Home View

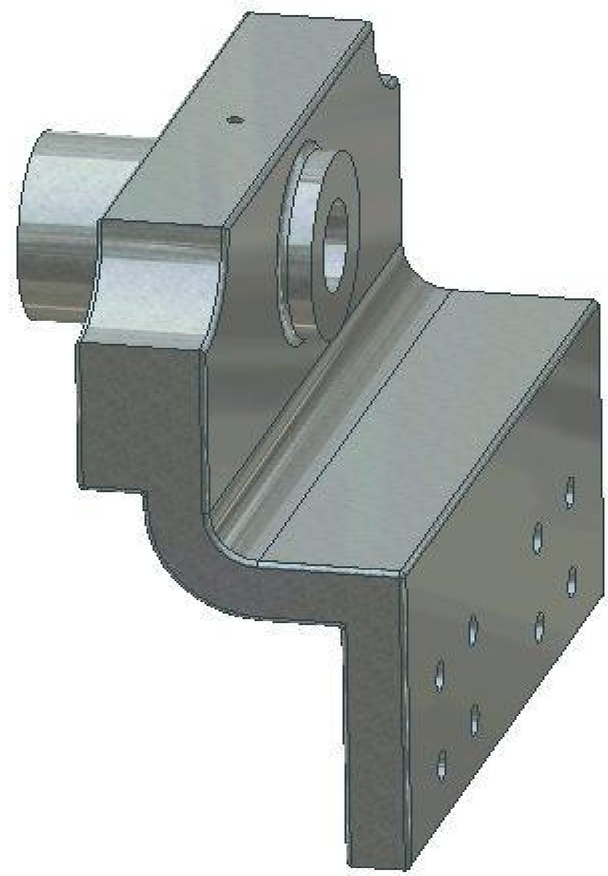


Figure Step 3B
Solid Model – Rotated View

Step 4

Create all fillets after the solid model is totally constructed.

Module 19 Work Features

Learning Outcomes

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

1. Describe and apply the POLYGON, TANGENT CIRCLE, and THREAD commands.
2. Describe work features including Work Points, Work Axes, and Work Planes and explain how they are used in model construction.
3. Describe and apply the WORK POINT, WORK AXIS, and WORK PLANE commands.

Geometry Lesson: Regular Polygons

A *polygon* is defined as any plane figure bounded by straight lines. A *regular polygon* is a polygon that has equal angles, equal sides and can be inscribed in or circumscribed around a circle. The first eight regular polygons are shown in Figure 19-1.

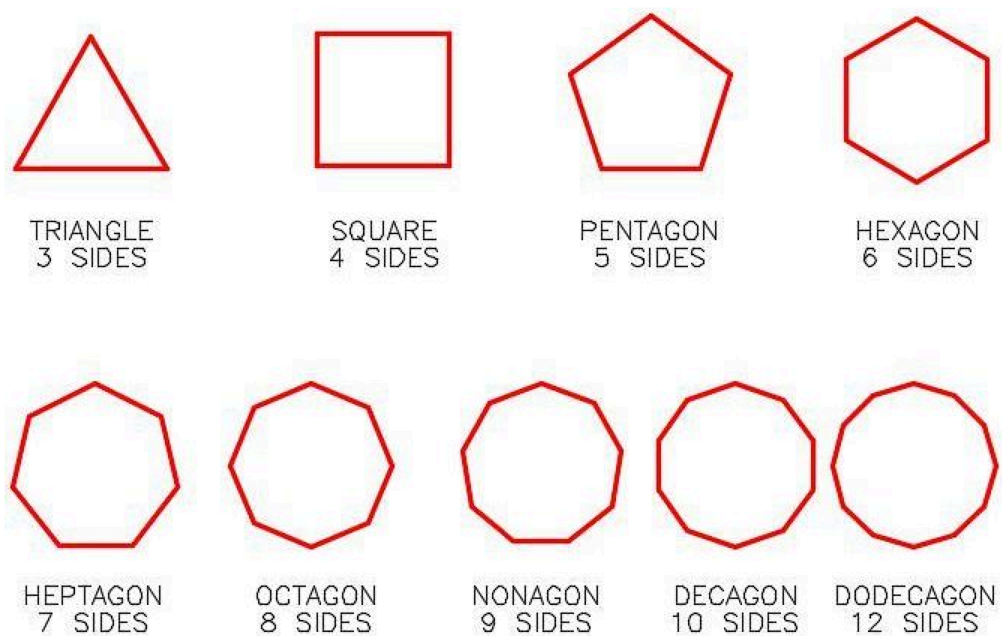


Figure 19-1
Regular Polygons
Work

Any regular polygon can be inscribed in or circumscribed around a circle as shown, using a hexagon, in Figure 19-2.

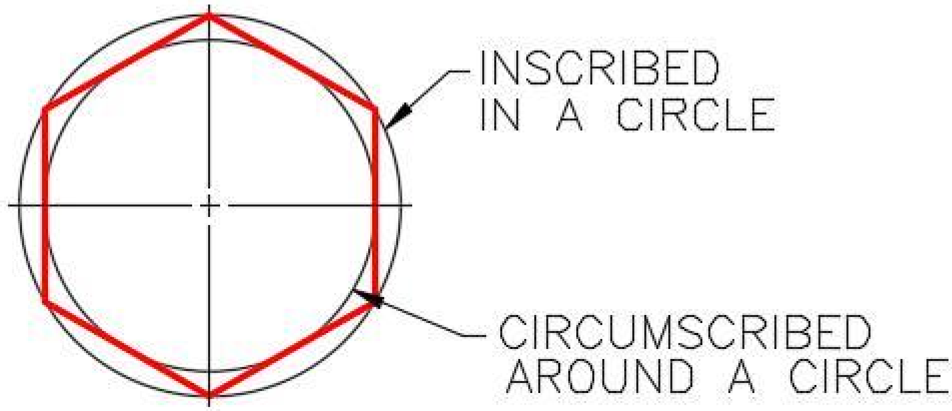


Figure 19-2
Inscribed and Circumscribed Regular Polygon

It is important to understand and know the geometry involved to construct a triangle, square, hexagon, and octagon as shown in Figure 19-3. Study each one and try to understand how they are constructed and the angles used to construct them.

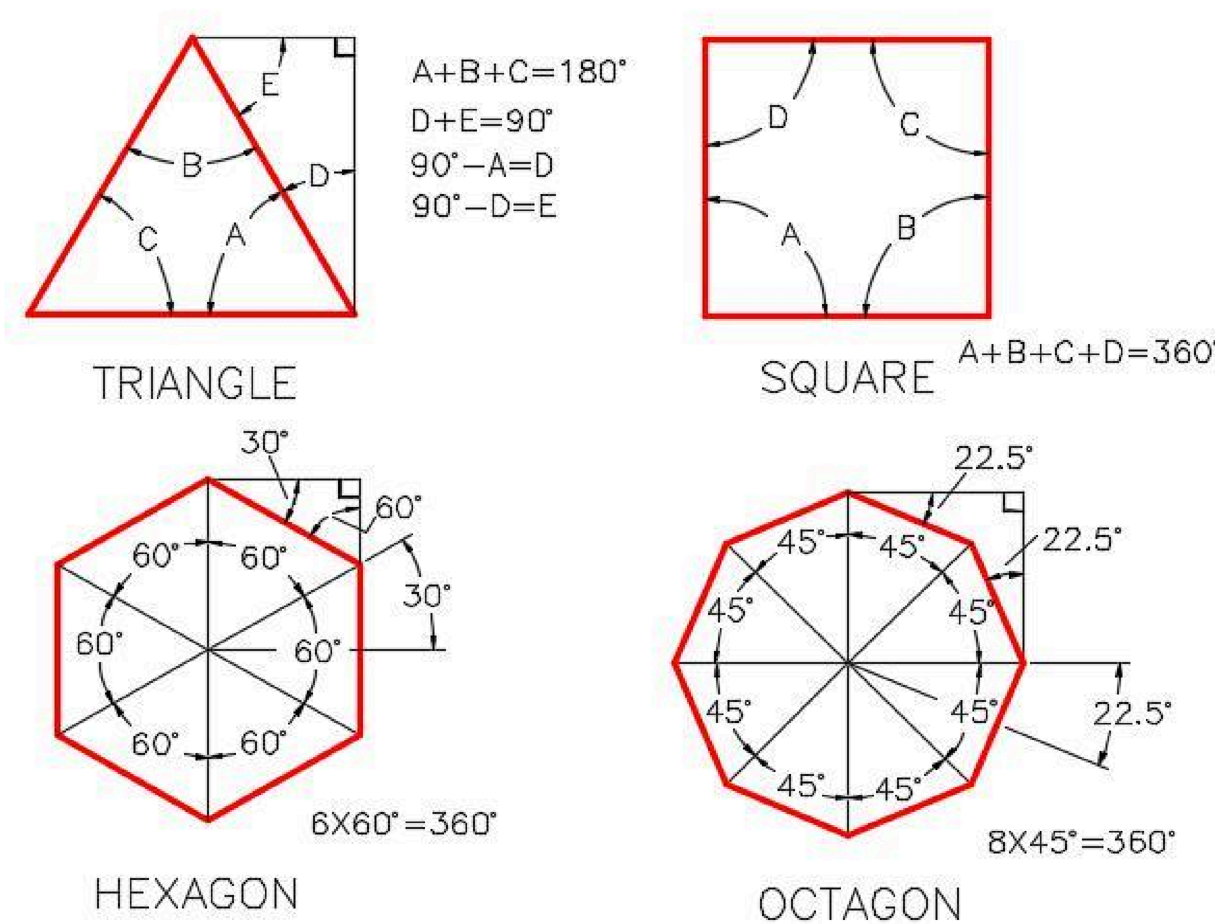
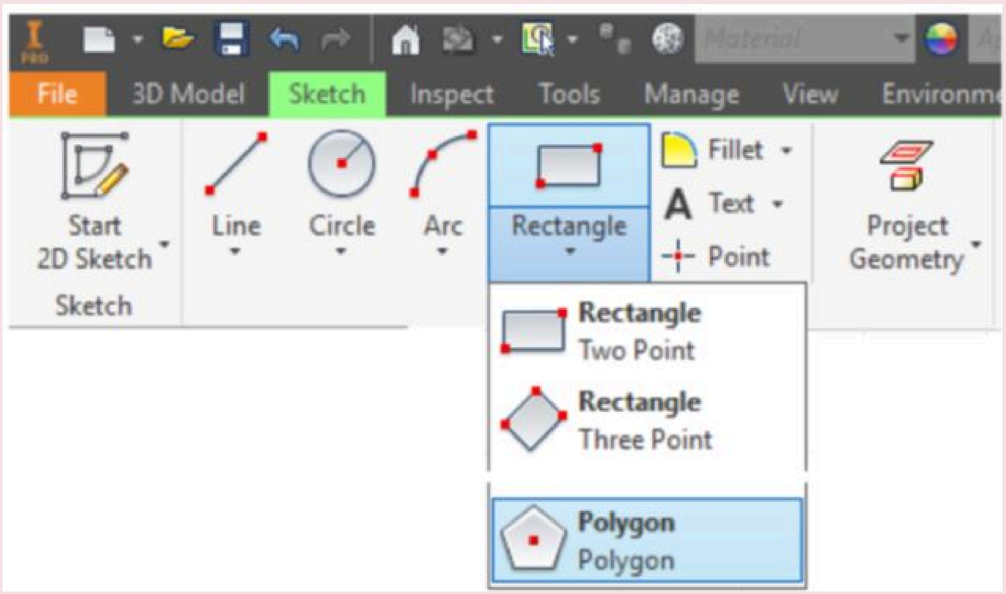


Figure 19-3
Geometry of Four Regular Polygons

Inventor Command: POLYGON

The POLYGON command is used to draw a regular polygon on a 2D sketch. You can select the number of sides and choose between either an inscribed or a circumscribed polygon.

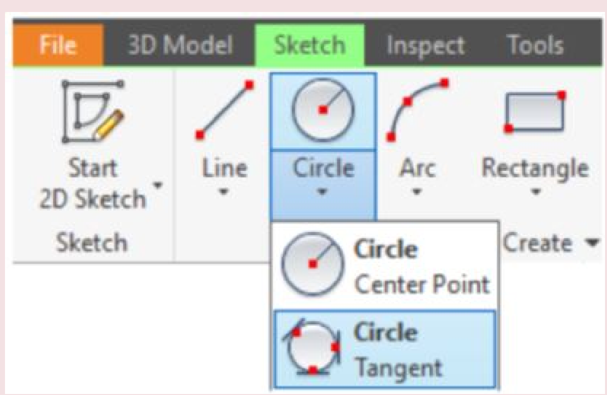
Shortcut: **none**



Inventor Command: TANGENT CIRCLE

The TANGENT CIRCLE command is used to draw a circle tangent to three lines.

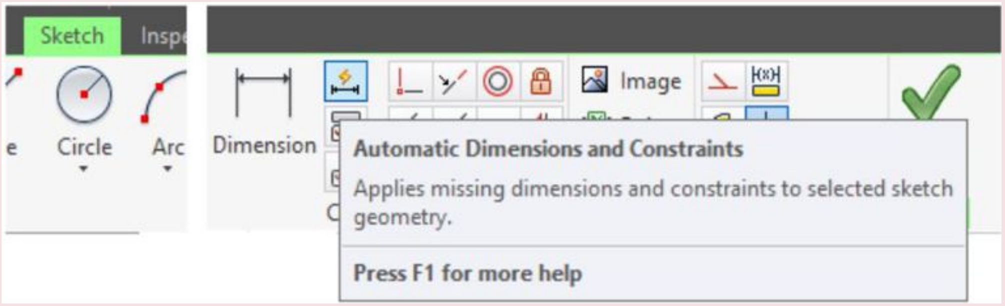
Shortcut: **none**



Inventor Command: AUTO DIMENSION

The AUTO DIMENSION command is used to add dimensions or constrains automatically to fully constrain a sketch.

Shortcut: **none**



Drawing Threads

The THREAD command is used to draw exterior threads or interior threads on a 3D solid model. See Figure 19-4 The threads created using the THREAD command are not actual threads constructed on the model. They are simply a graphical representation of the threads. A real life solid model created from the Inventor part would not be threaded. Actual threads can be created but this is a much more advanced feature that is taught in the Inventor Advanced book. The thread specifications can be applied to the thread in the sketch and then be used when creating the working drawing of the part.

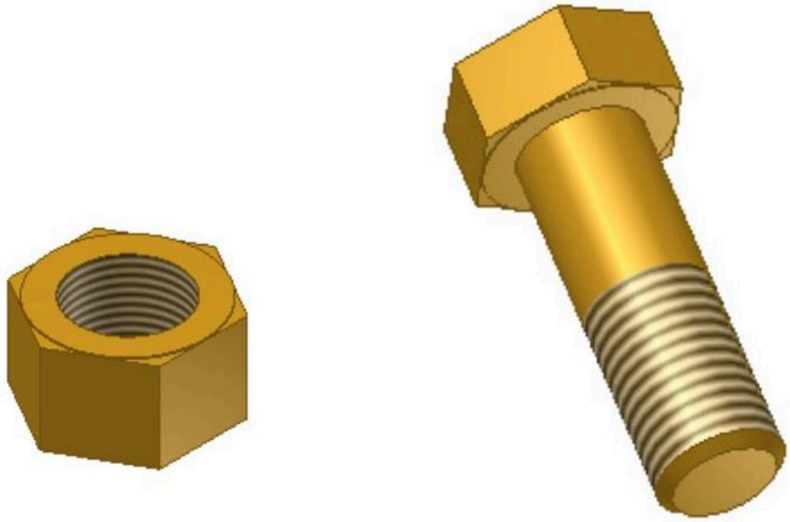
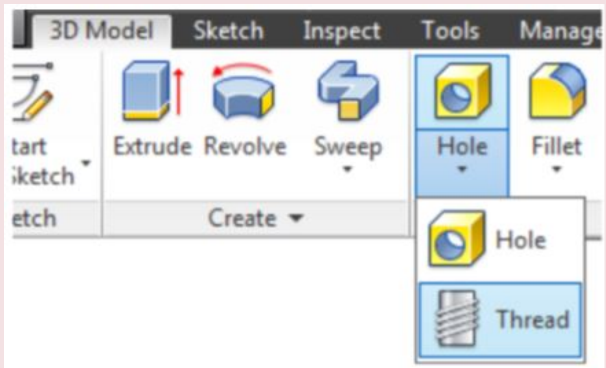


Figure 19-4
External and Internal Threads

Inventor Command: THREAD

The THREAD command is used to draw a graphical representation of an outside or inside diameter thread. The thread specifications can also added to the thread properties.

Shortcut: **none**



WORK ALONG: Drawing Polygons and Threads – Part 1

Step 1

Using the NEW command start a new part file using the template: English-Modules Part (in).ipt.

Step 2

Save the file with the name: Inventor Workalong 19-1. (Figure Step 2A, 2B, and 2C)

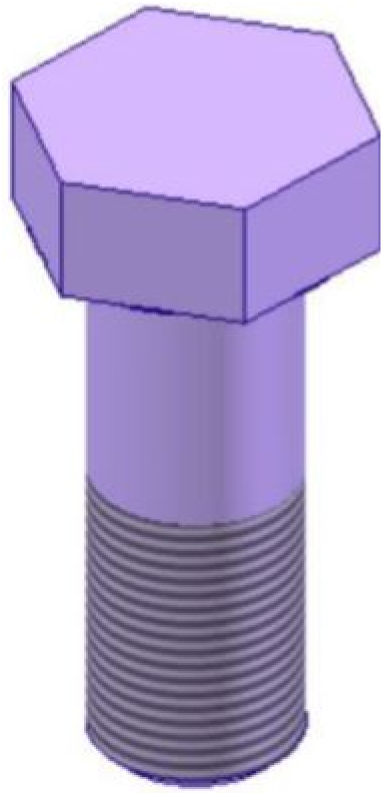


Figure Step 2A
3D Model –
Home View

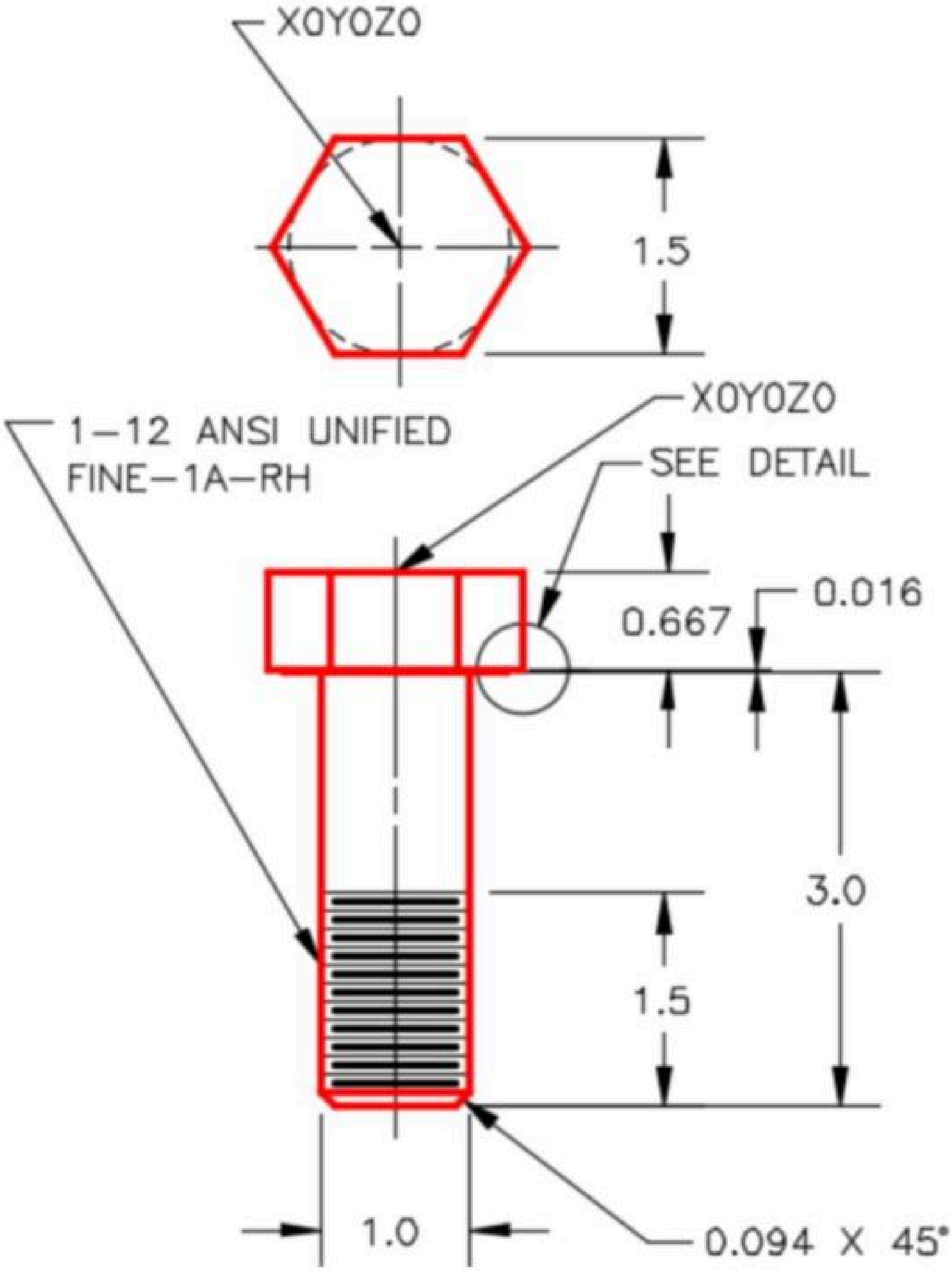


Figure Step 2B
Dimensioned Multiview Drawing

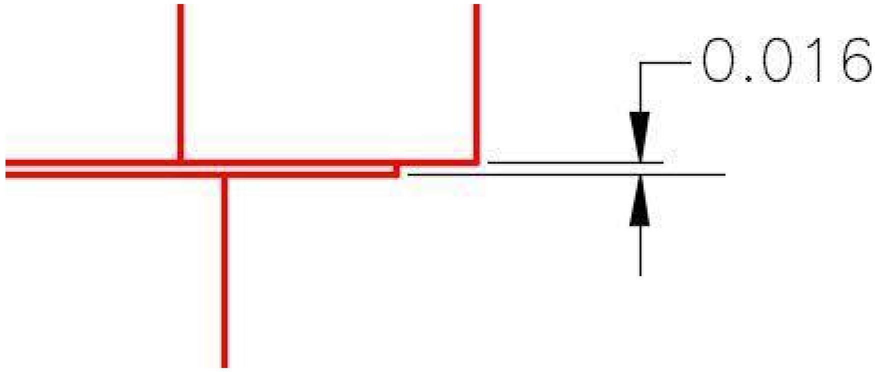


Figure Step 2C Detail

Step 3

Draw the Base sketch on the Top or XY plane.

Step 4

Project the Center Point onto the Base sketch.

Step 5

Enter the POLYGON command and set the number of sides to 6. (Figure Step 5)

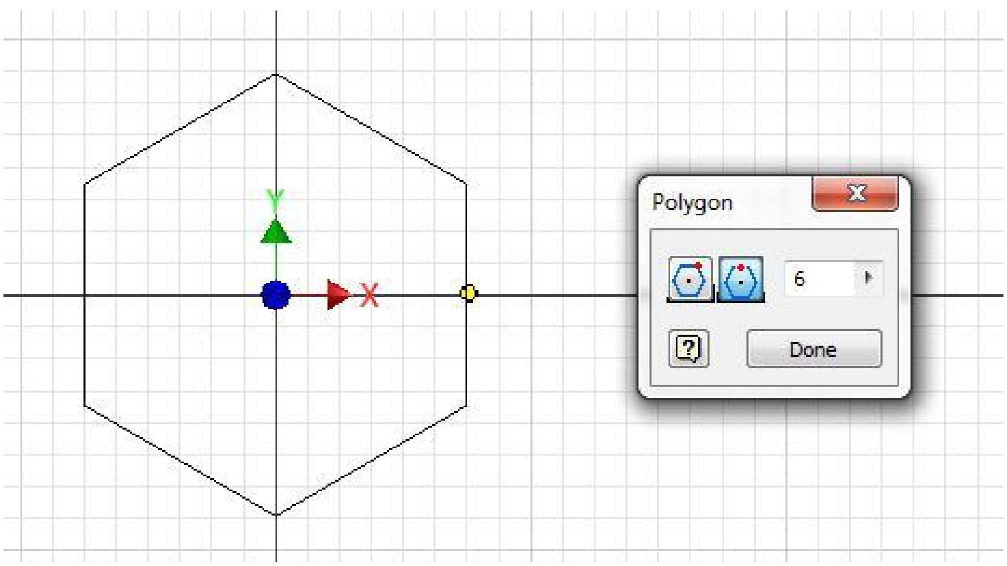


Figure Step 5

Step 6

Enable the Circumscribed Polygon icon. Snap to the Center Point for the centre and select a radius of approximately 0.75 inches. (Figure Step 6)

Fig Step 6

AUTHOR'S COMMENTS: A circumscribed polygon is used in this sketch because the edge to edge dimension was given rather then corner to corner.

Step 7

Add the dimension from the Center Point to the left line. (Figure Step 7)

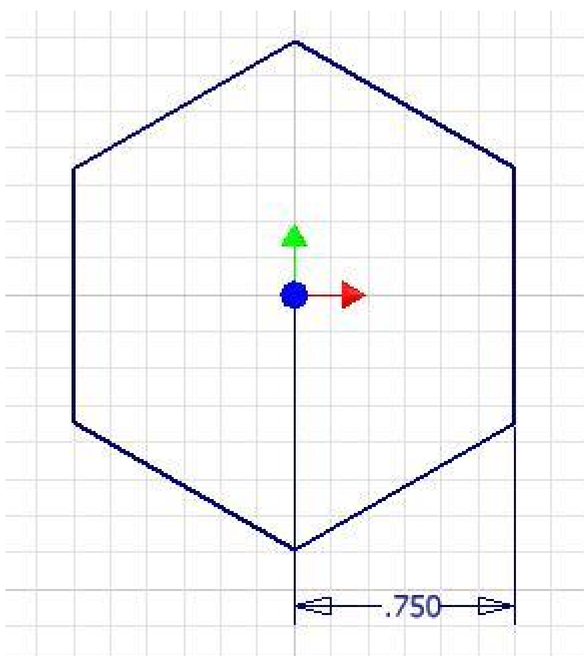


Figure Step 7

Step 8

Enter the AUTO DIMENSION command. Note that one dimension or constraint is required to fully constrain the sketch. Click Apply. (Figure Step 8A, 8B, and 8C

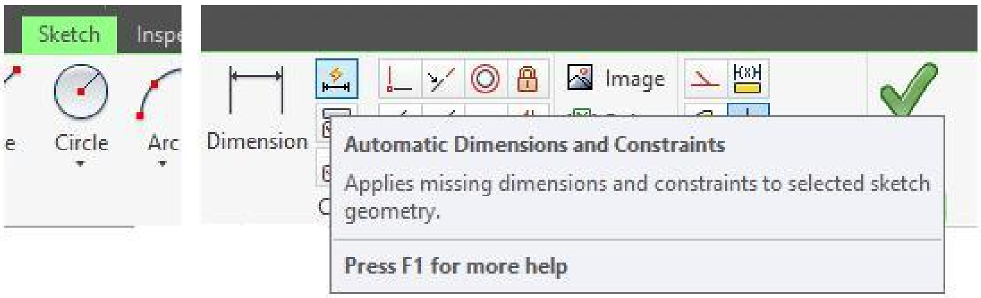


Figure Step 8A

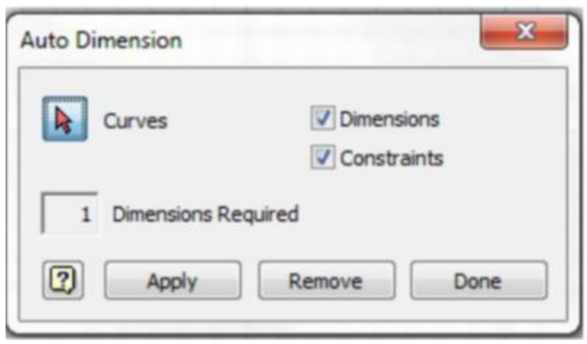


Figure Step 8B

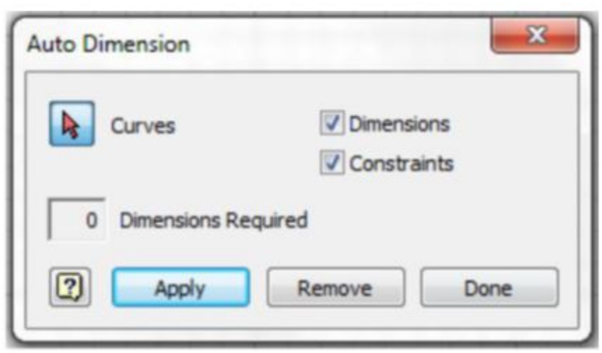


Figure Step 8C

AUTHOR'S COMMENTS: This will add a constraint to fully constrain the sketch. The AUTO DIMENSION command is used to automatically add the dimensions or constraints to fully constrain it.

Step 9

Extrude the hexagon in the negative Z direction. (Figure Step 9)

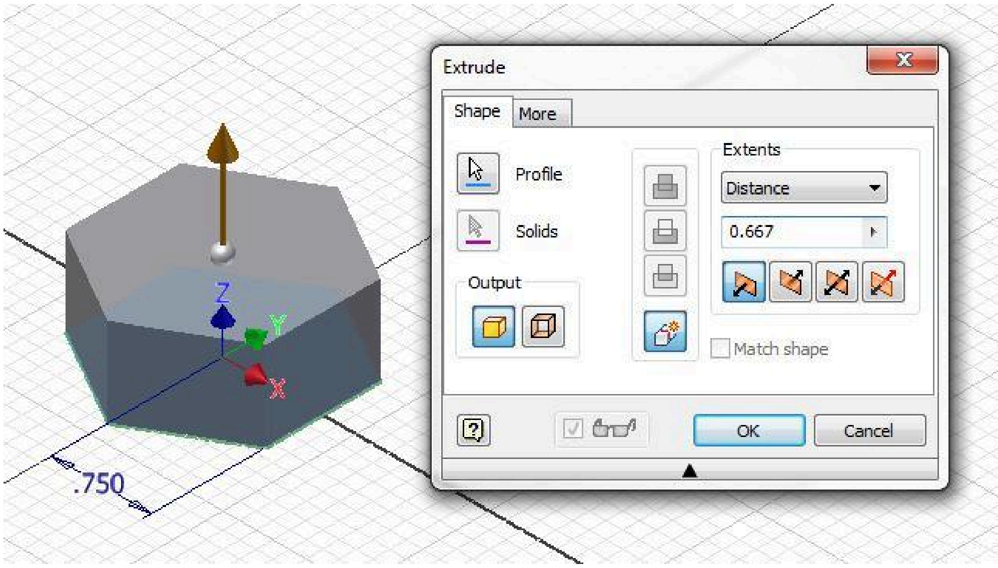


Figure Step 9 [Click to see image full size]

Step 10

Start a new sketch on the bottom side of the Base model. Enter the TANGENT CIRCLE command. When prompted, select any three edges. (Figure Step 10)

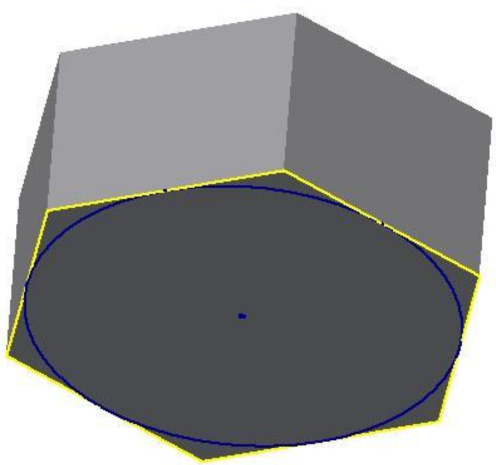


Figure Step 10

AUTHOR'S COMMENTS: Since you are inserting a circle tangent inside of a regular polygon, any three edges can be selected.

Step 11

Extrude the circle 0.016 inches. Ensure that you enable the Join icon and extrude it away from the bolt head. (Figure Step 11)

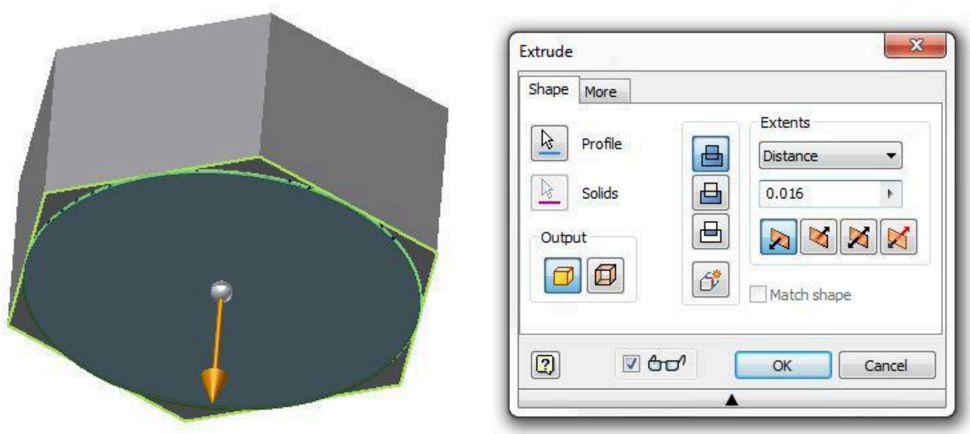


Figure Step 11 [Click to see image full size]

Step 12

Start a new sketch on the extruded circle. Draw and dimension a 1 inch circle using the centre of the circle to snap to. Extrude the circle 3 inches. (Figure Step 12A and 12B)

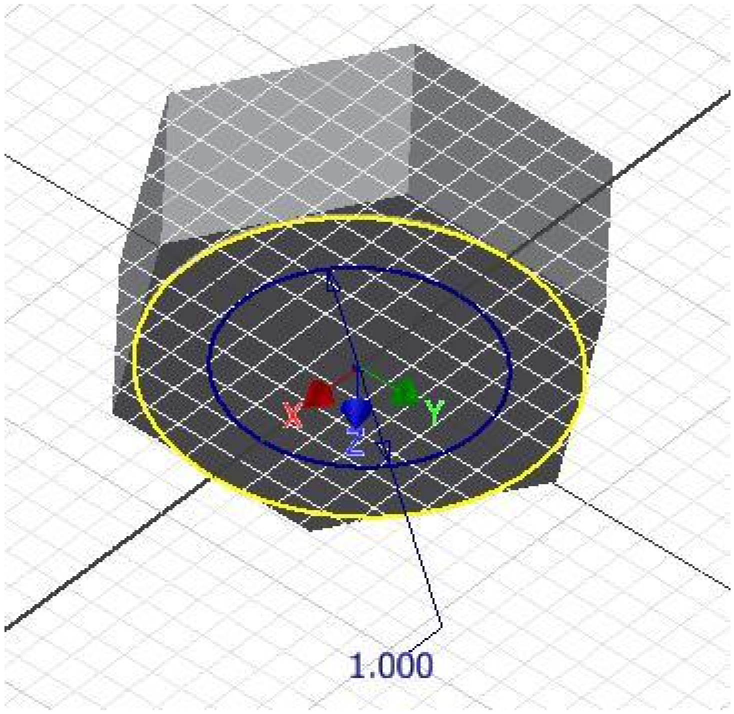


Figure Step 12A

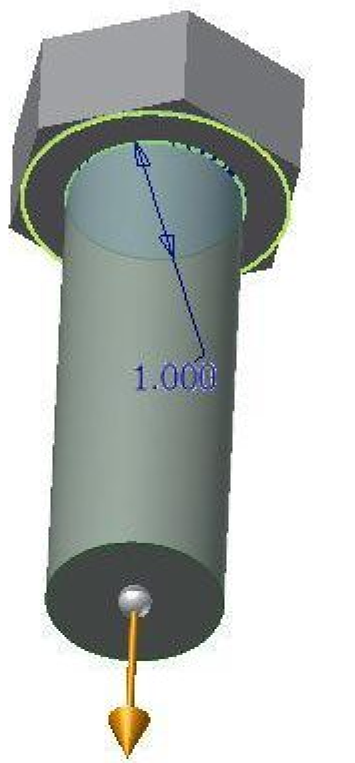


Figure Step 12B

Step 13

Using the CHAMFER command, chamfer the bottom of the bolt. (Figure Step 13)

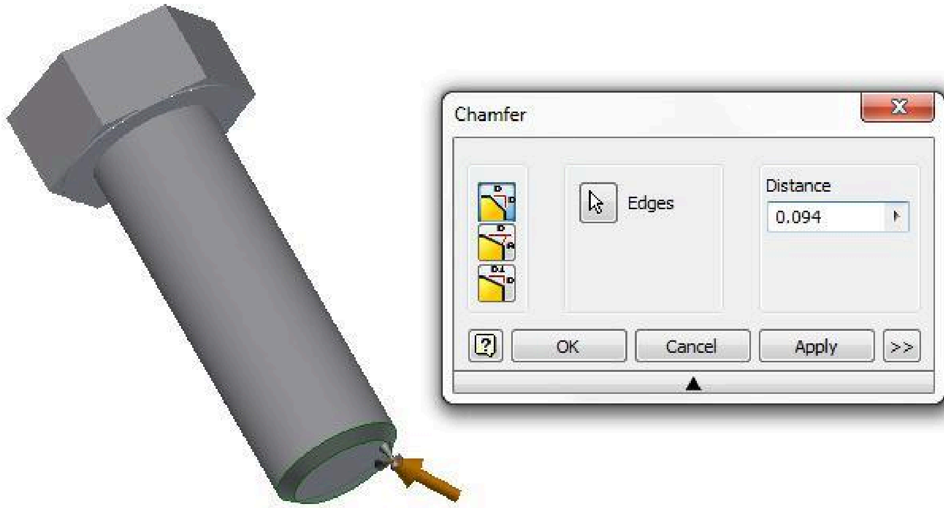


Figure Step 13 [Click to see image full size]

Step 14

Enter the THREAD command. Enable the Location tab and set the Length to 1.5 and then select the cylinder as the face to thread. Do not close the dialogue box. (Figure Step 14)

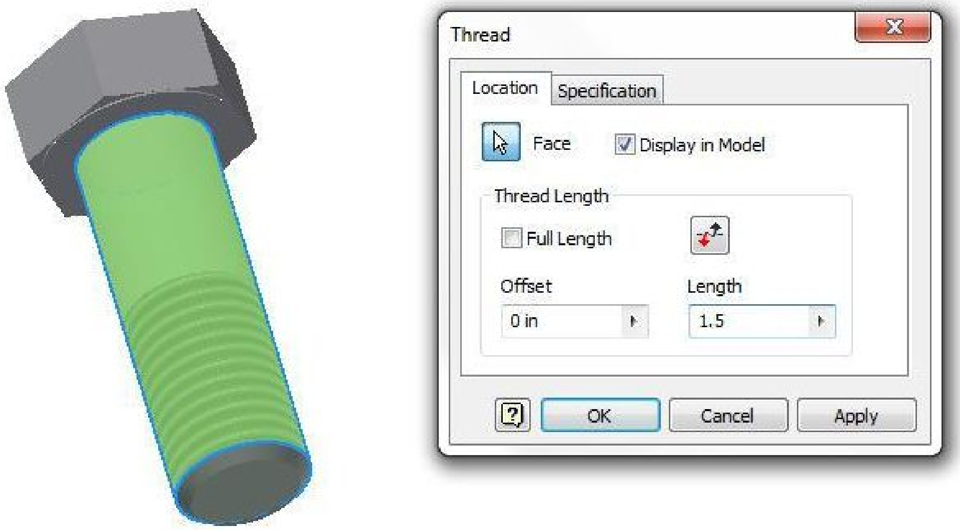


Figure Step 14 [Click to see image full size]

Step 15

Enable the Specification tab. Set the thread specifications as shown in the figure. Click OK to execute the command. (Figure Step 15A and 15B)

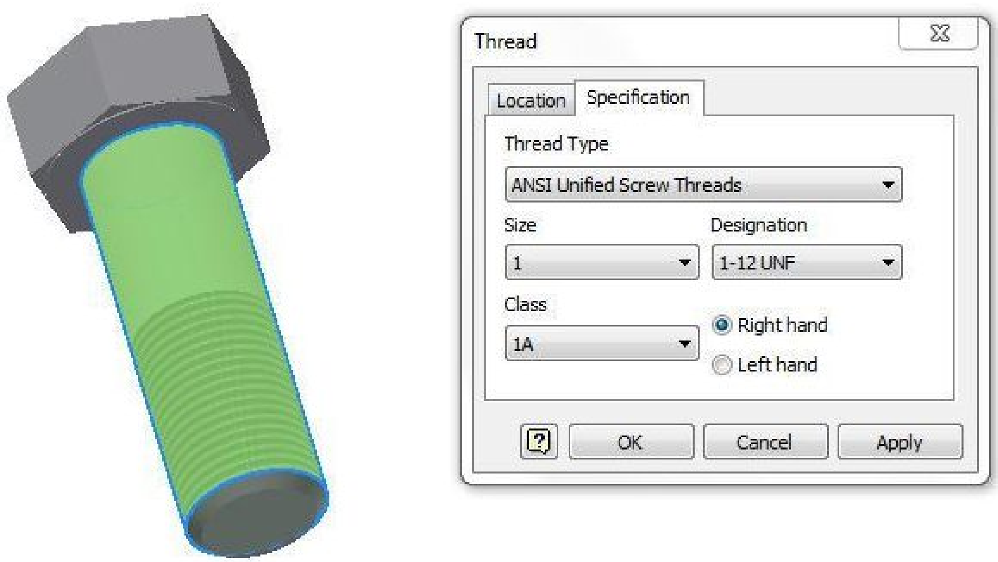


Figure Step 15A [Click to see image full size]

Figure Step 15B

Step 16

Change the view to the Home view and change the colour to: Metal-Brass. (Figure Step 16)

Figure Step 16

Step 17

Save and close the file.

> **MUST KNOW:** The threads created by the THREAD command are not actual threads constructed on the model. They are simply a graphical representation of the threads. A real life solid model created from the Inventor part file would not be threaded.

WORK ALONG: Drawing Polygons and Threads – Part 2

Step 1

Check the default project and if necessary, set it to Inventor Course.

Step 2

Using the NEW command start a new part file using the template: English-Modules Part (in).ipt.

Step 3

Save the file with the name: Inventor Workalong 19-2.

Step 4

Using what you learned in the last workalong, construct the nut shown in the multiview drawing. (Figure Step 4A and 4B)

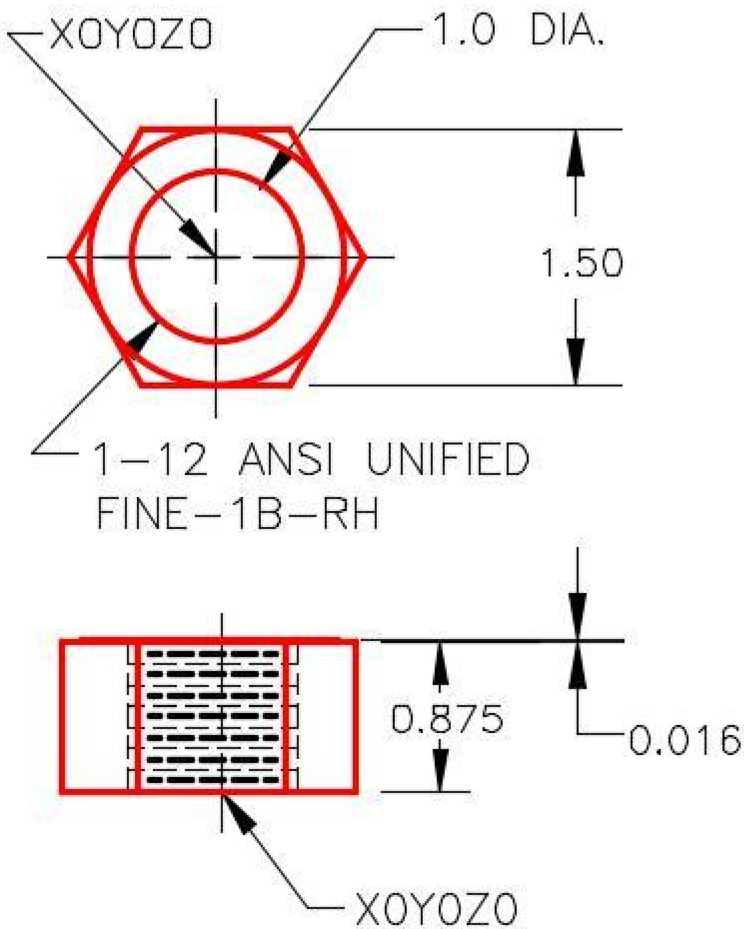


Figure Step 4A
Dimensioned Multiview Drawing

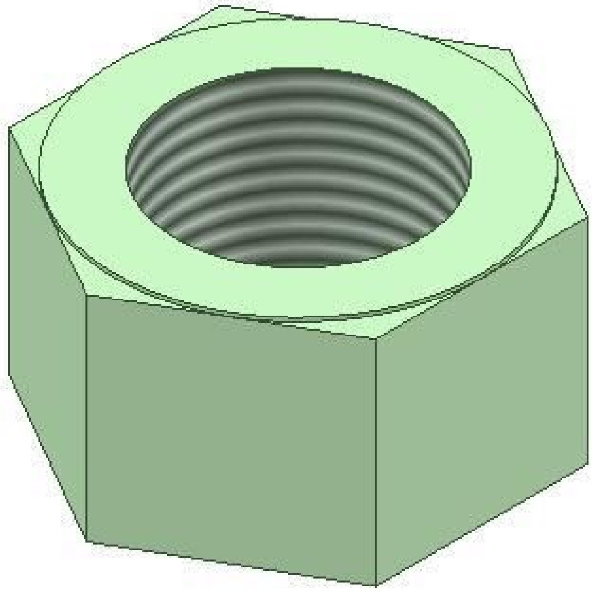


Figure Step 4B 3D Model – Home View

Step 5

Change the colour to: Metal-Brass. (Figure Step 5A and 5B)

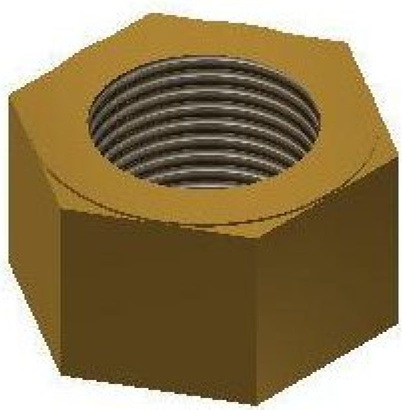


Figure Step 5A

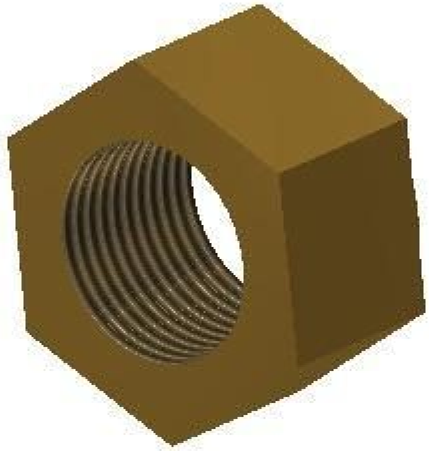


Figure Step 5B

Work Features

Work Features consists of points, axes, and planes that can be inserted on a solid model or in Model space. They are used as a construction aids to draw the model. You can insert them when existing geometry cannot be used to add geometry. The three work features available in Inventor are the work point, the work axis, and the work plane. See Figure 19-5. This module is an introduction to work features only and will teach the user how to create and use them in part modeling. They are also used in assemblies.

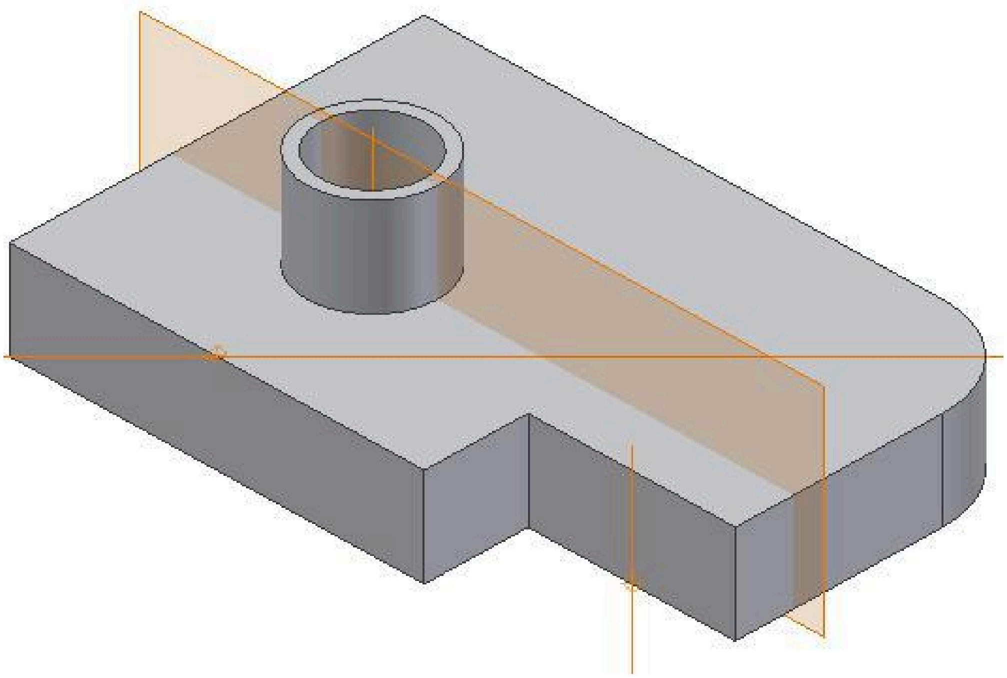


Figure 19-5
Work Features

Work Points

A *Work Point* is a parametric construction point or a single XYZ location that is inserted and then used as a work feature. Once a Work Point is inserted on the model, it can be projected onto part faces, linear edges, or onto an arc or circle using the PROJECT GEOMETRY command. Work Points can also be constrained to the centre points of arcs, circles, and ellipses. In this module, inserting Work Points on the model or in 3D space will be taught.

Work Axes

A *Work Axis* is a parametric construction line or two XYZ locations joined by a line that is inserted and then used as a work feature. Even though a Work Axis appears as a specified length, as far as Inventor is concerned, it is infinite in length and can be expanded to any length required.

Use Work Axes when creating models to mark symmetry lines, centre lines, or distances between revolved feature axes. Work Axes can be used along the symmetry lines of circular features such as cylinders, shafts, or holes. They can also be created as a work axis on a linear edge, a sketch line, or a 3D sketch line.

Work Planes

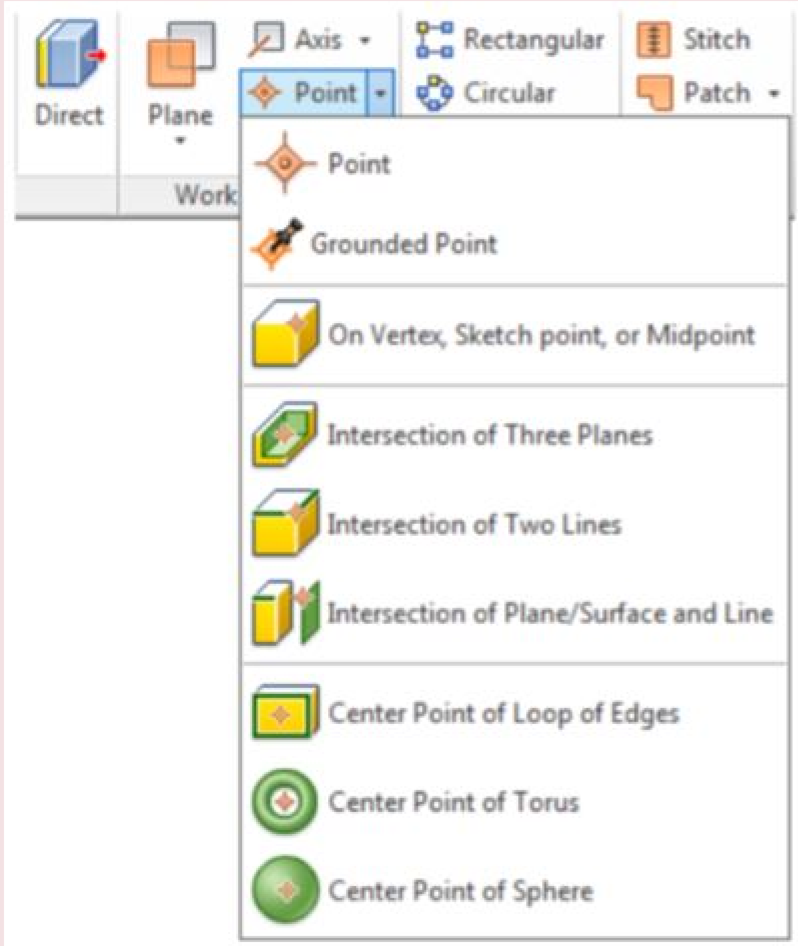
A *Work Plane* is parametric construction plane or four XYZ locations joined by lines inserted on the model or in model space and then used as a work feature. Even though a Work Plane appears as a rectangular plane of a given size, it is actually infinite in size and can be expanded to any size required.

Work Planes can be placed at any orientation in space, offset from existing faces, or rotated around an axis or edge. A work plane can be used as a sketch plane and dimensioned or constrained to other features or components. Each work plane has its own internal coordinate system. The order in which geometry is selected determines the origin and positive directions of the coordinate system axes.

Inventor Command: WORK POINT

The WORK POINT command is used to insert a Work Point on the model or in Model space.

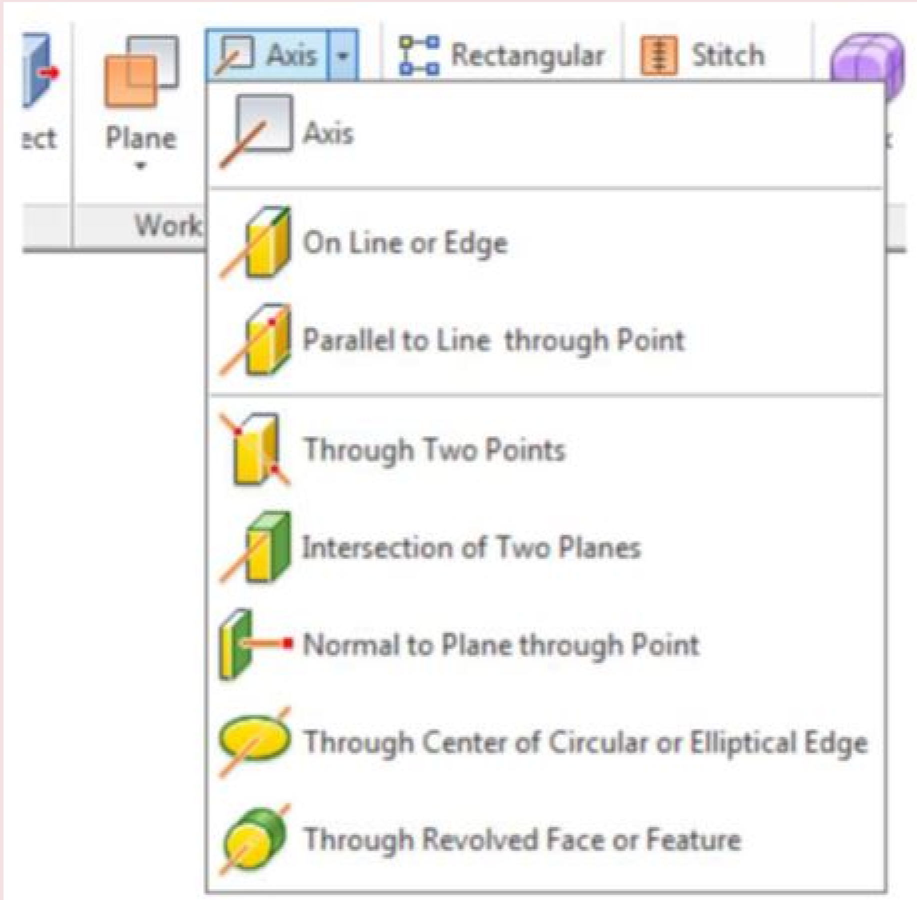
Shortcut: **. (period)**



Inventor Command: WORK AXIS

The WORK AXIS command is used to insert a Work Axis on the model or in Model space.

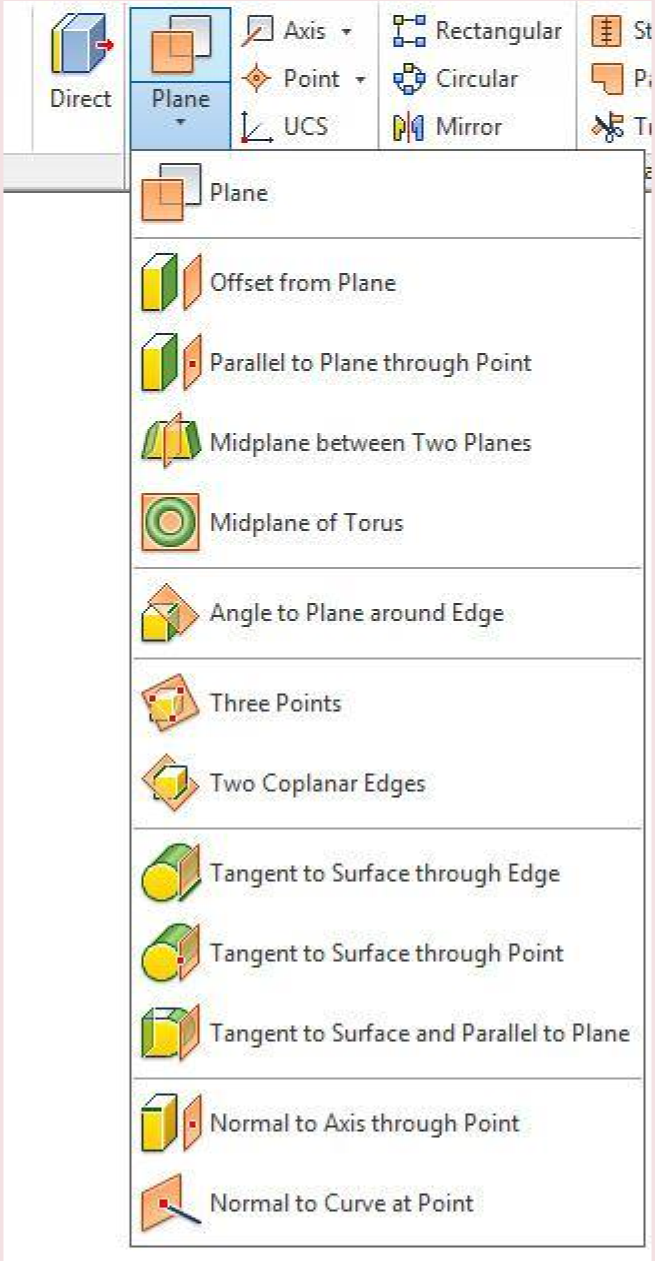
Shortcut: **/ (forward slash)**



Inventor Command: WORK PLANE

The WORK PLANE command is used to insert a Work Plane on the model or in Model space.

Shortcut:**] (right bracket)**



WORK ALONG: Creating Work Features

Step 1

Check the default project and if necessary, set it to Inventor Course.

Step 2

Using the NEW command start a new part file using the template: English-Modules Part (in).ipt.

Step 3

Save the file with the name: Inventor Workalong 19-3.

Step 4

Create the solid model shown in the figures. (Figure Step 4A and 4B)

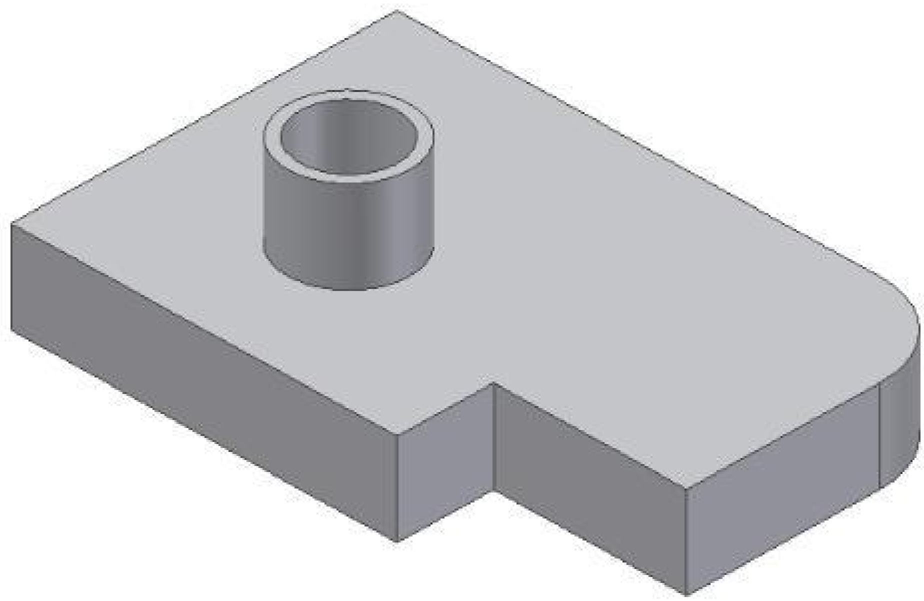


Figure Step 4A
3D Model – Home View

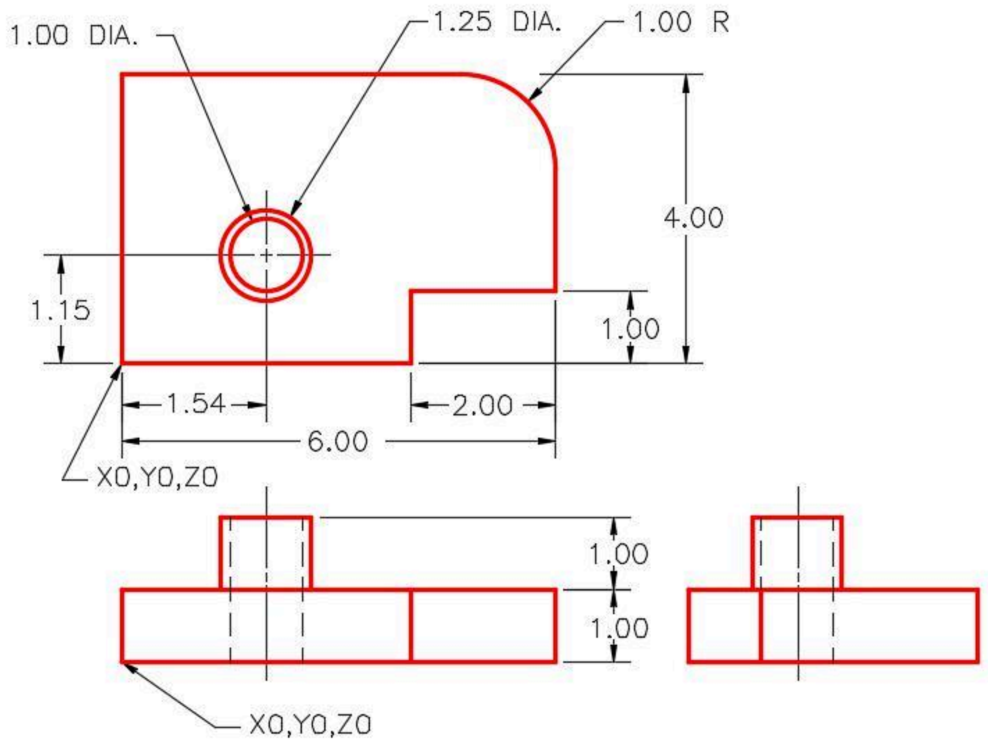


Figure Step 4B
Dimensioned Multiview Drawing [Click to see image full size]

Step 5

Insert a Work Point at the apparent intersection of two edges. Click the WORK POINT command and then Intersection of Two Lines. Select the edges shown in the figure. Note how they highlight and extend. (Figure Step 5A, 5B, and 5C)

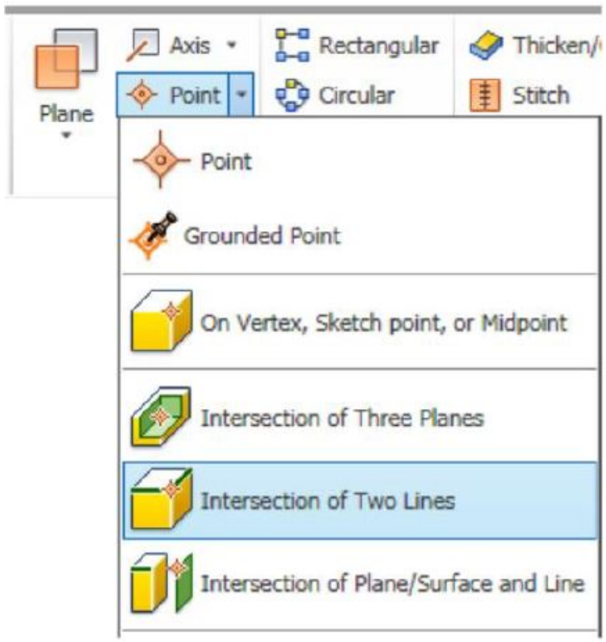


Figure Step 5A

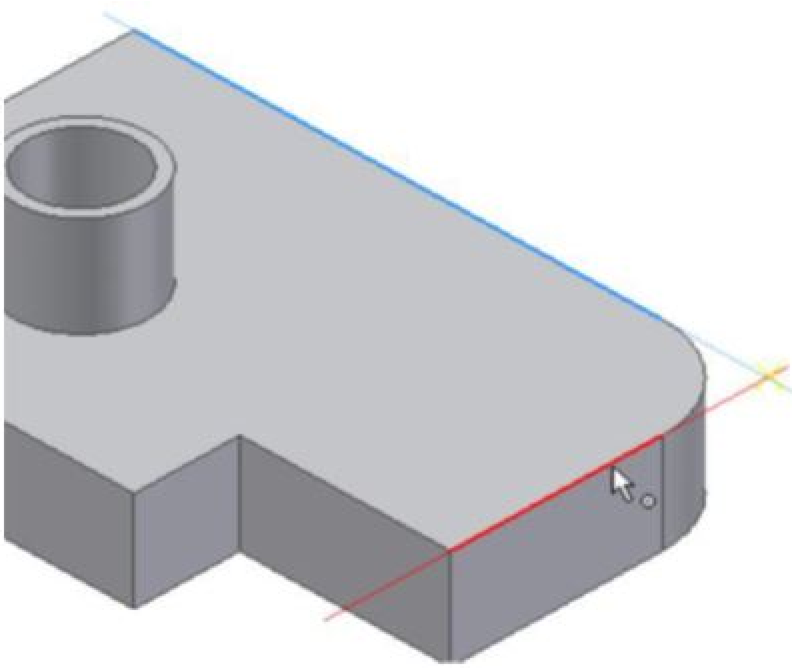


Figure Step 5B

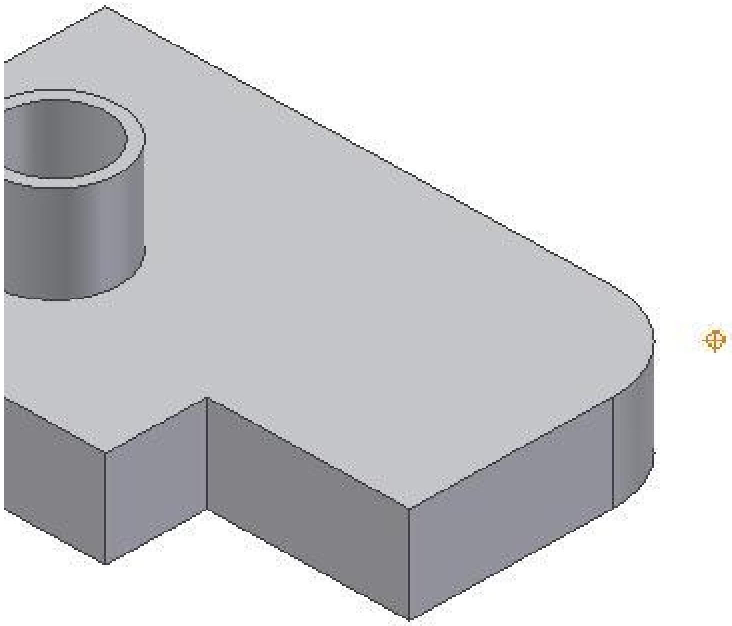


Figure Step 5C

AUTHOR'S COMMENTS: The Work Point will be inserted at the apparent intersection of the two selected edges.

Step 6

Insert a Work Point at the intersection of three planes. Click the WORK POINT command and then Intersection of Three Planes. Select the planes shown in the figures. (Figure Step 6A, 6B, 6C, 6D, and 6E)

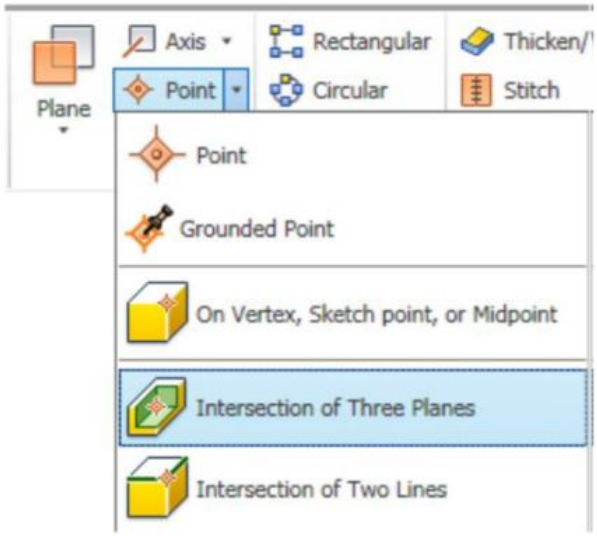


Figure Step 6A

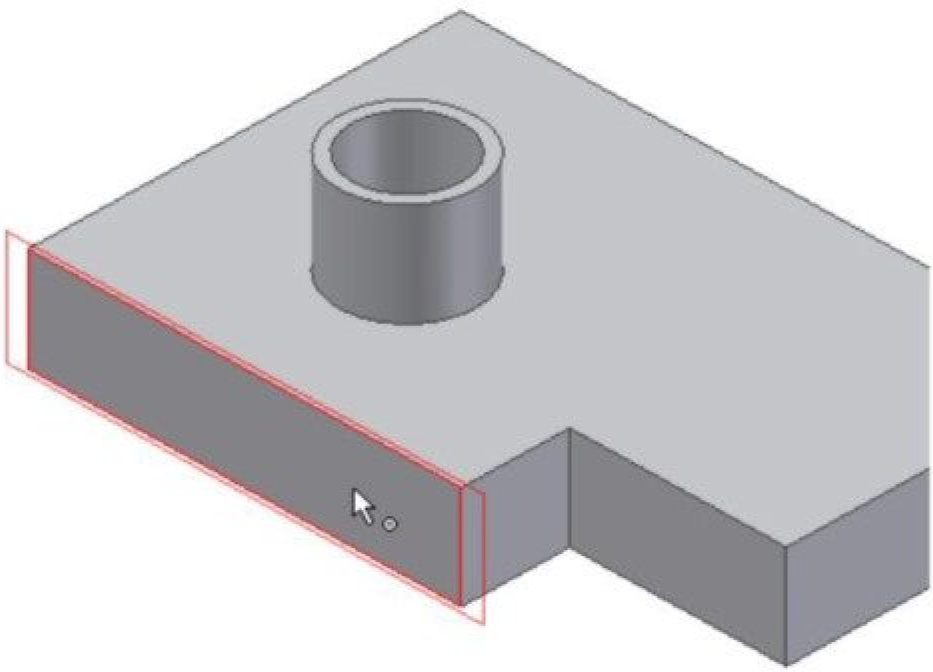


Figure Step 6B

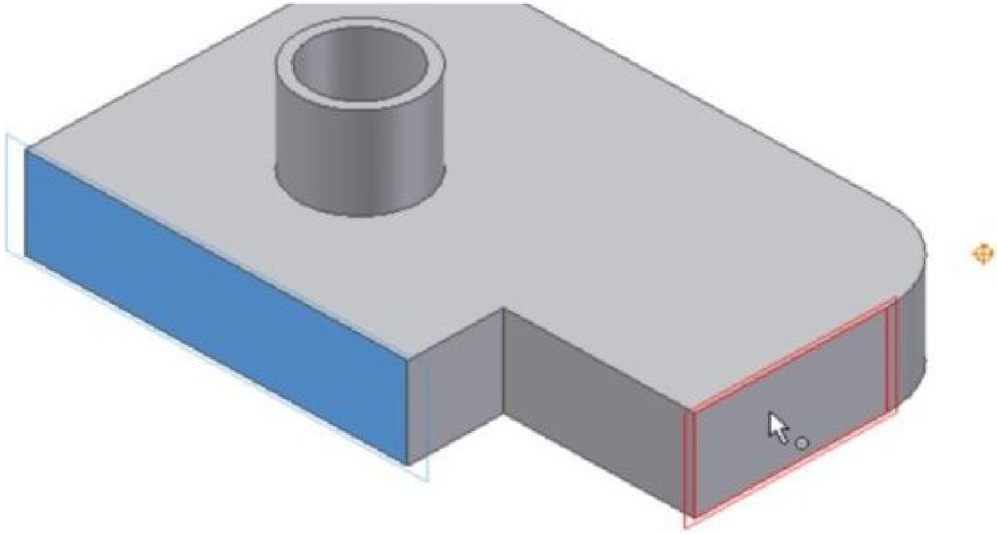


Figure Step 6C

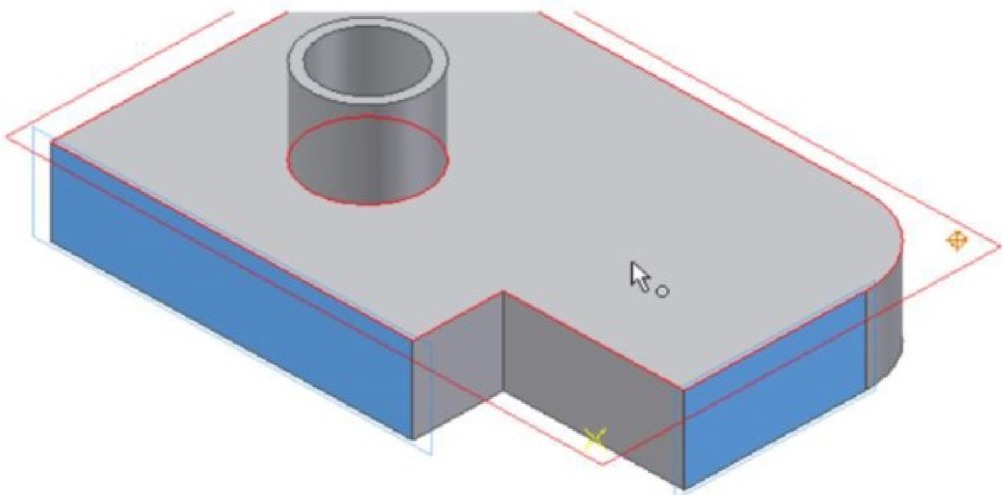


Figure Step 6D

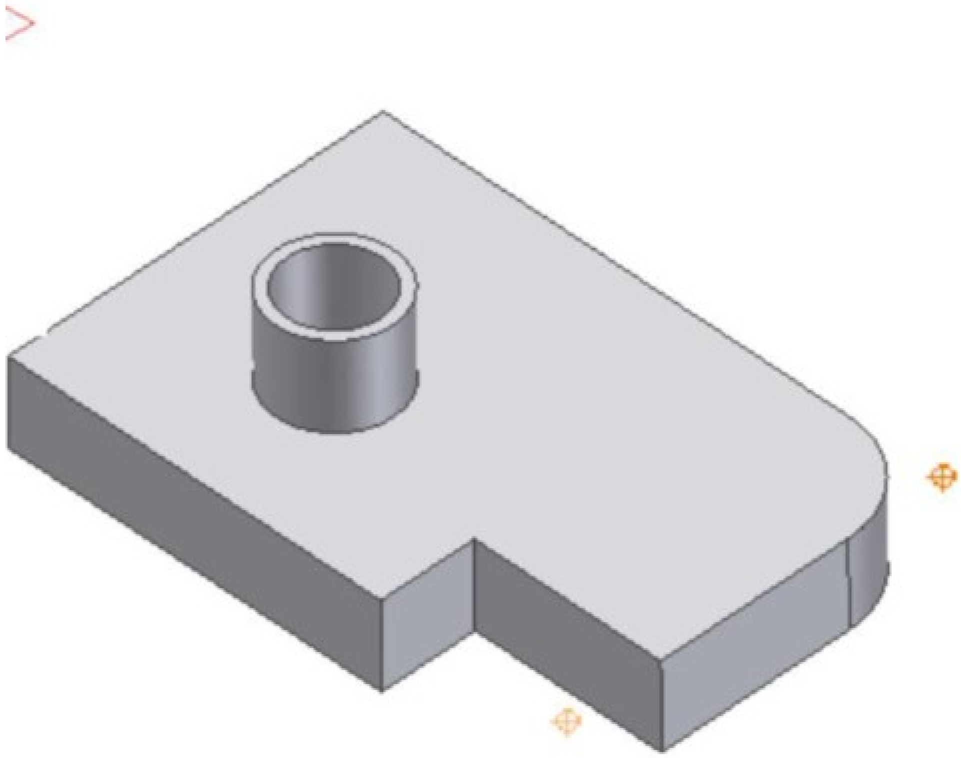


Figure Step 6E

Step 7

To insert a Work Point at the midpoint of an edge. Click the WORK POINT command and then On Vertex, Sketch point, or Midpoint. Snap to the midpoint of the edge shown in the figure. The midpoint will highlight with a yellow work point symbol. Select it when it appears. (Figure Step 7A, 7B, and 7C)

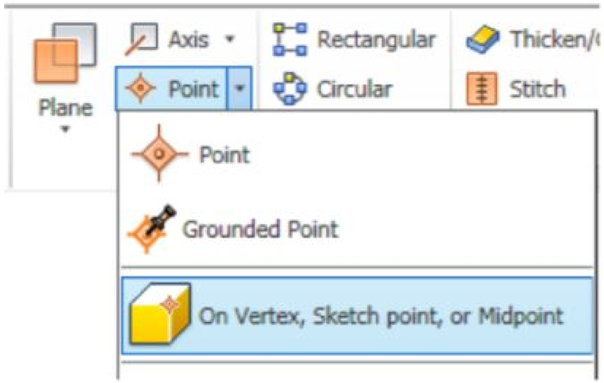


Figure Step 7A

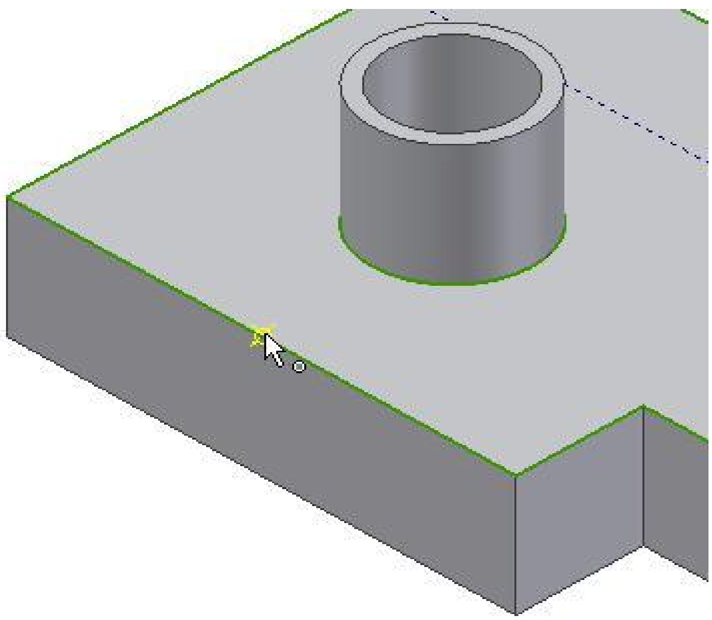


Figure Step 7B

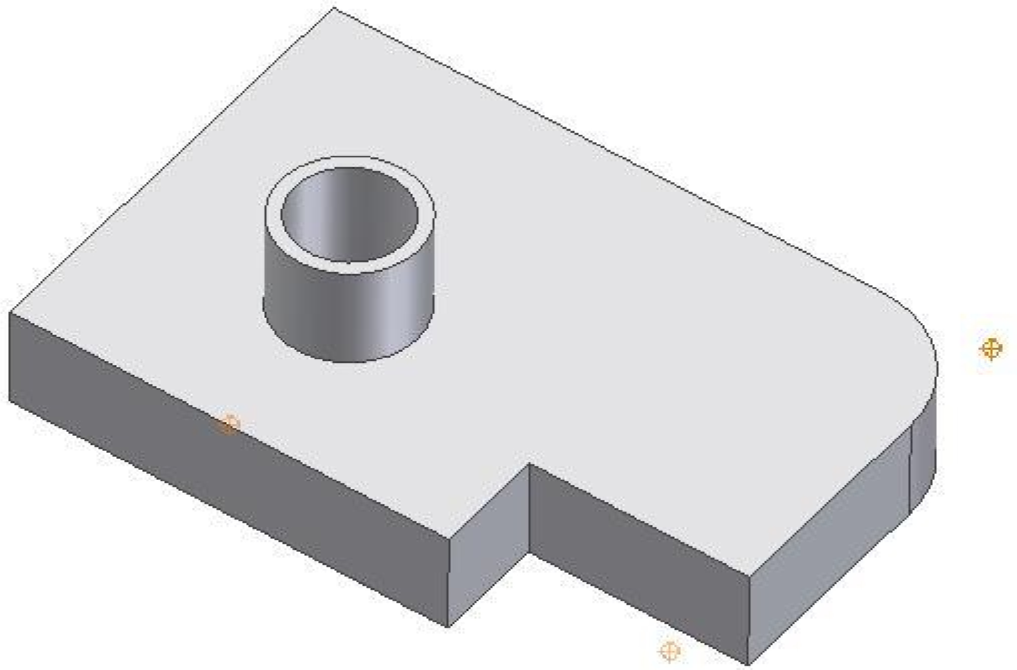


Figure Step 7C

AUTHOR'S COMMENTS: You should now have three Work Points inserted.

Step 8

Work Features will display as items in the Browser bar. Note the three Work Points that you just inserted. The visibility of Work Features can be enabled or disabled in the Right-click menu. (Figure 8A and 8B)

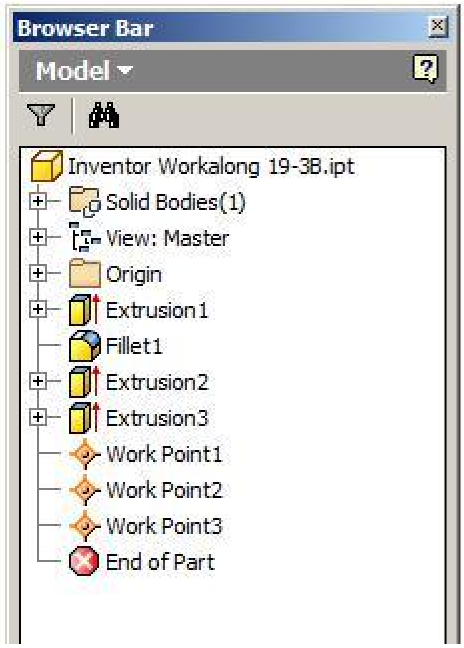


Figure Step 8A

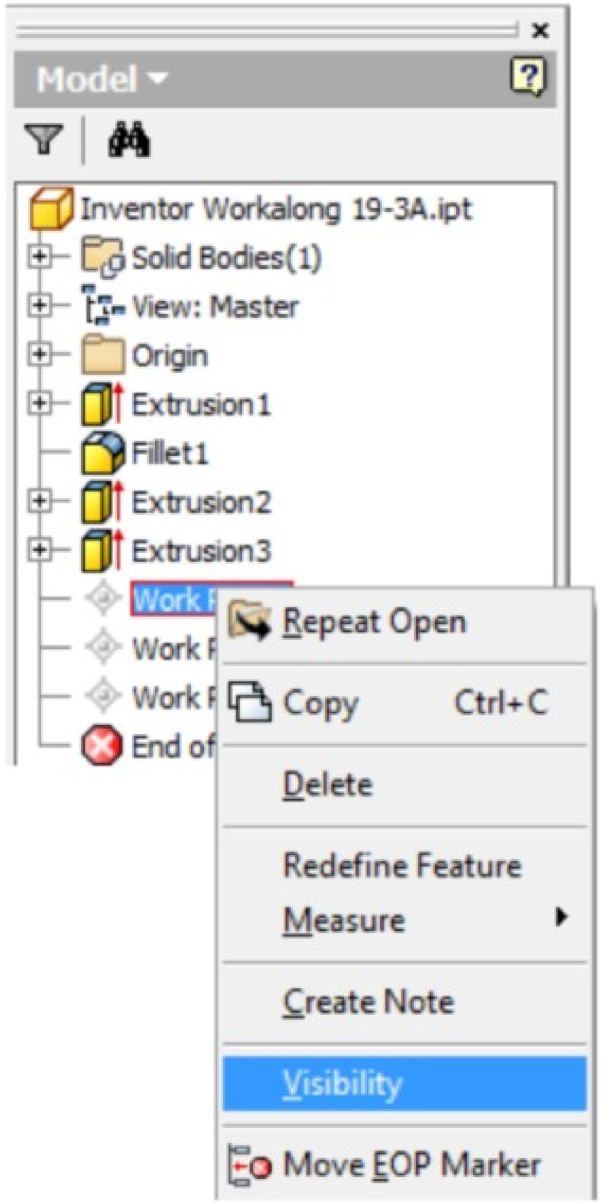


Figure Step 8B

Step 9

Insert a Work Axis on the centerline of the cylinder. Enter the WORK AXIS command and then Through Center of Circular or Elliptical Edge. Select the cylinder as shown in the figure. Note how the cylinder highlights. The axis will appear as a colored line. (Figure Step 9A, 9B, and 9C)

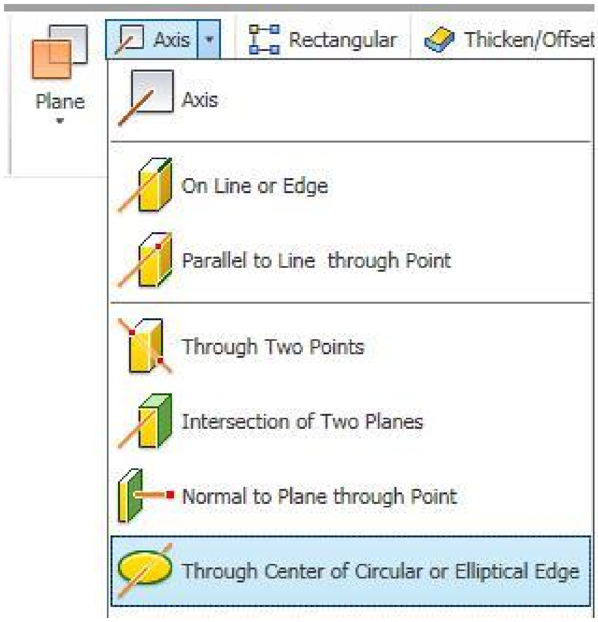


Figure Step 9A

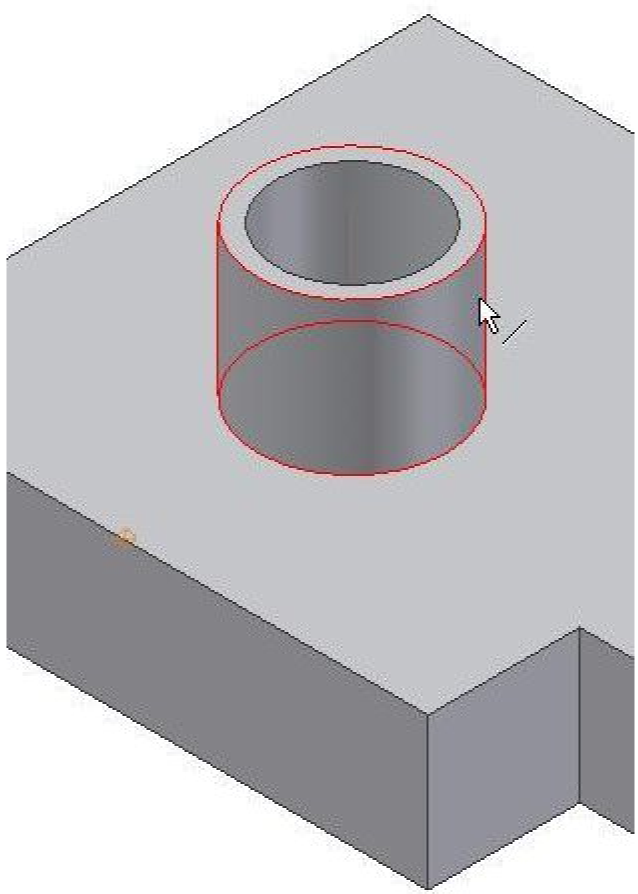


Figure Step 9B

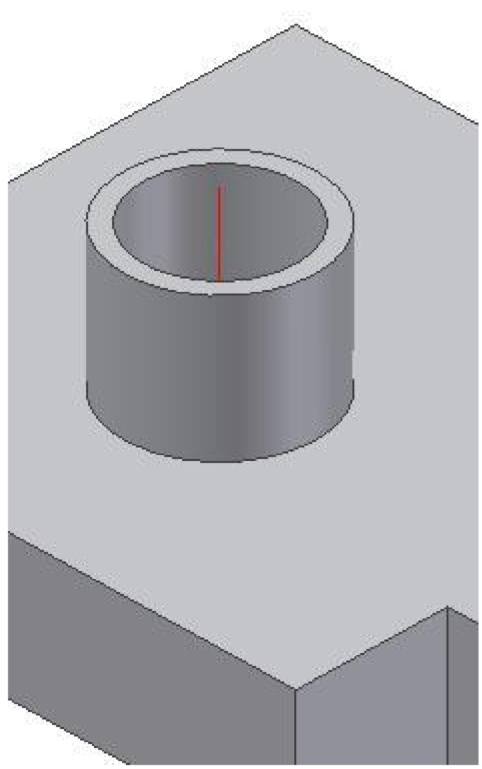


Figure Step 9C

Step 10

Insert a Work Axis between two Work Points. Enter the WORK AXIS command and then Through Two Points. Select the Work Points on the front edge and the back corner. (Figure Step 10A, 10B, 10C, and 10D)

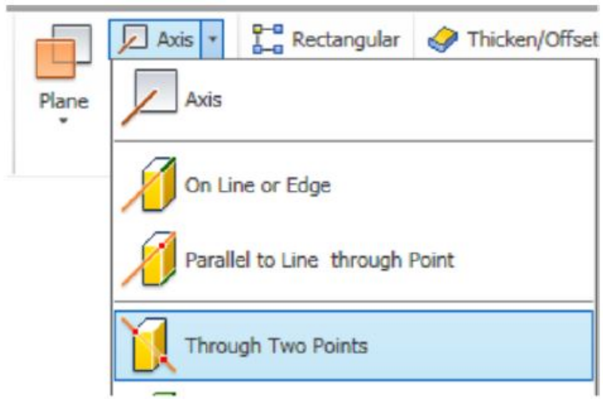


Figure Step 10A

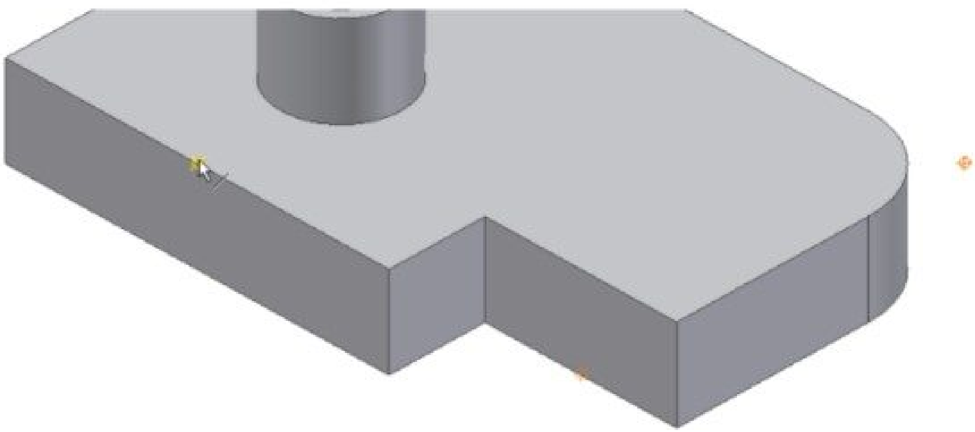


Figure Step 10B

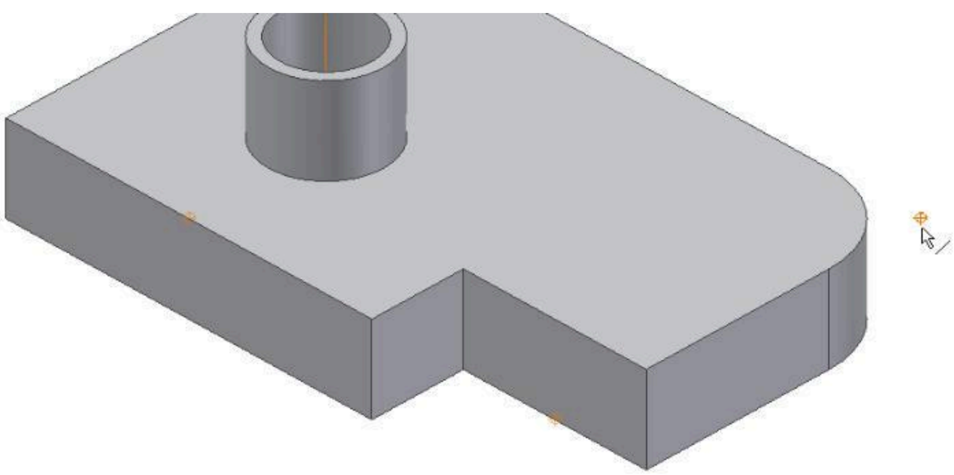


Figure Step 10C

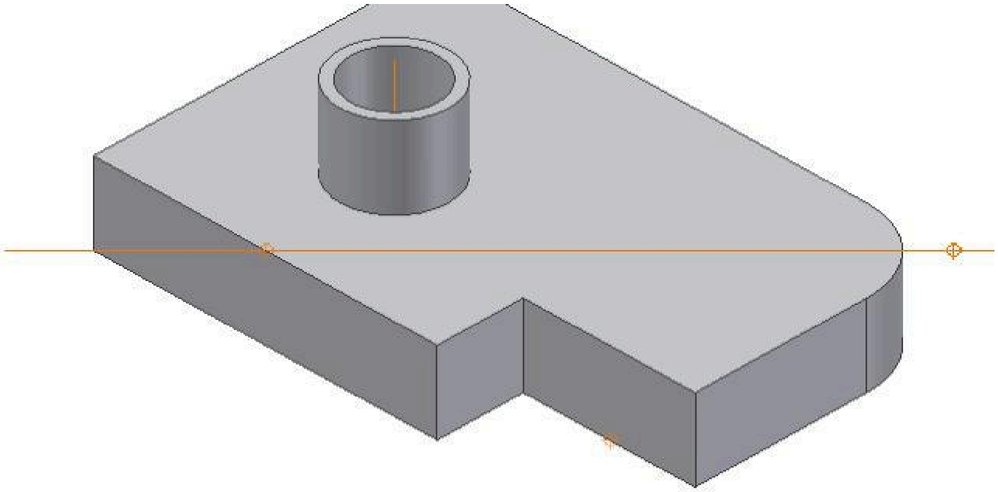


Figure Step 10D

Step 11

Insert a Work Axis at the intersection of two non-parallel faces. Enter the WORK AXIS command and then Intersection of Two Planes. Select the face on the left side and then the face on the right side. The Work Axis will appear at the apparent intersection of the two faces. (Figure Step 11A, 11B, 11C, and 11D)

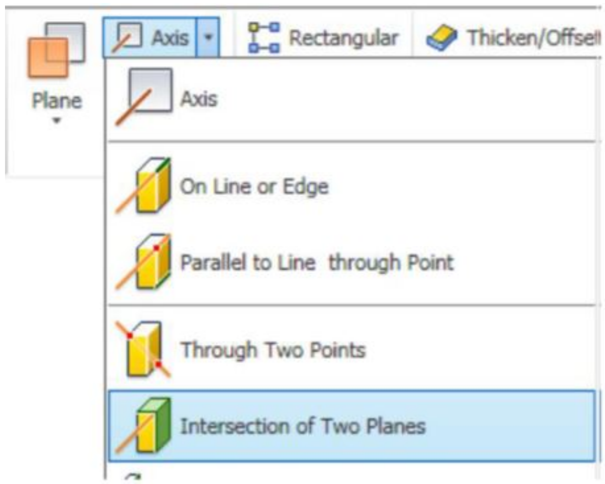


Figure Step 11A

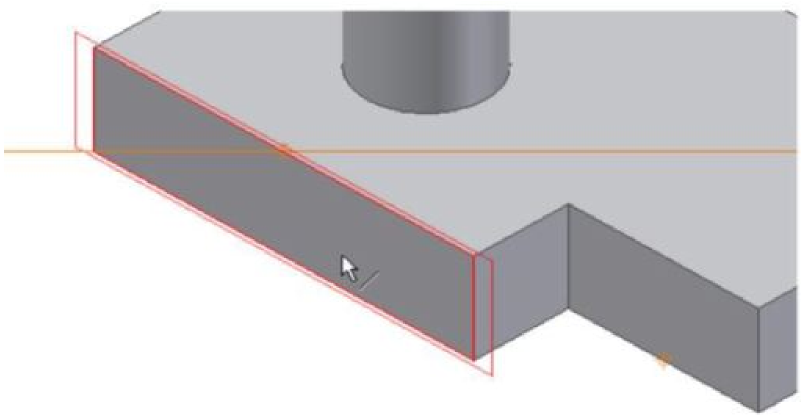


Figure Step 11B

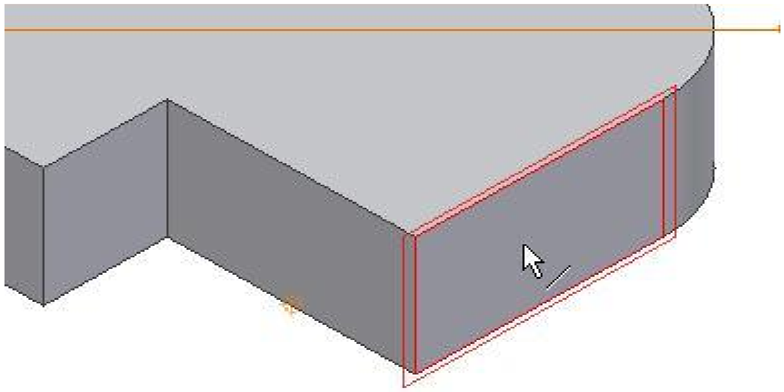


Figure Step 11C

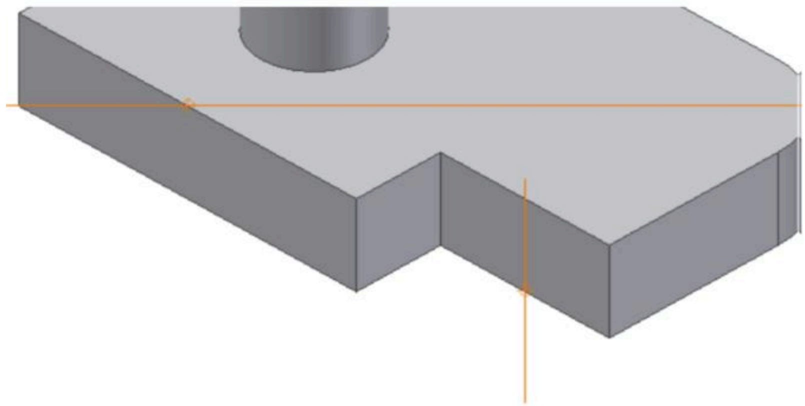


Figure Step 11D

Step 12

Work Features will display as items in the Browser bar. Note the three Work Axes that you just inserted. (Figure Step 12)

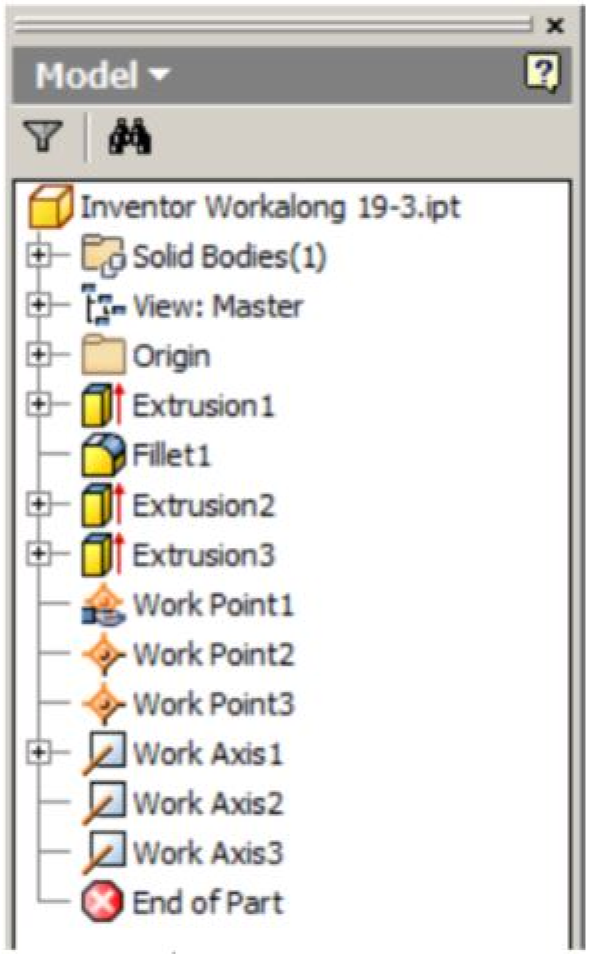


Figure Step 12

Step 13

Insert a Work Plane from an existing face and through a Work Axis at a specified angle. Enter the WORK PLANE command and then Plane. Select the face on the right side of the model and then select the Work Axis at the centre of the cylinder. Enter the angle between the new Work Plane and the face. In this case, 90 degrees or perpendicular. (Figure Step 13A, 13B, 13C, and 13D)

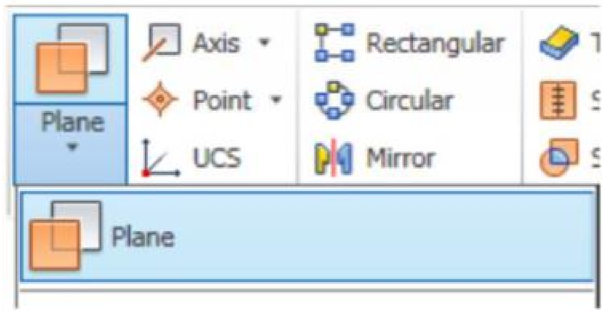


Figure Step 13A

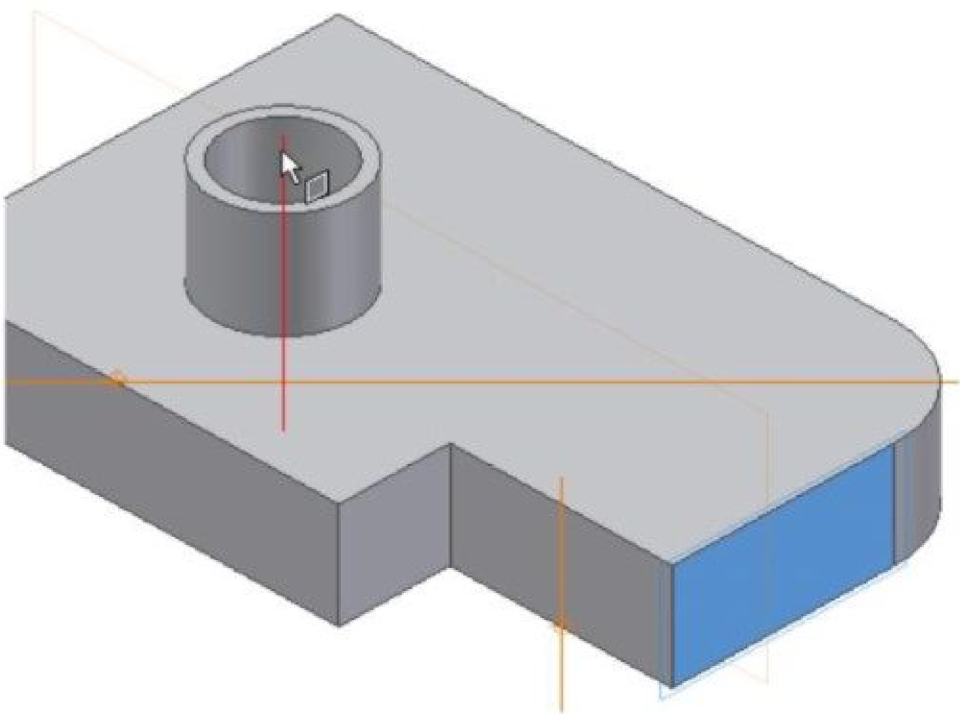


Figure Step 13B

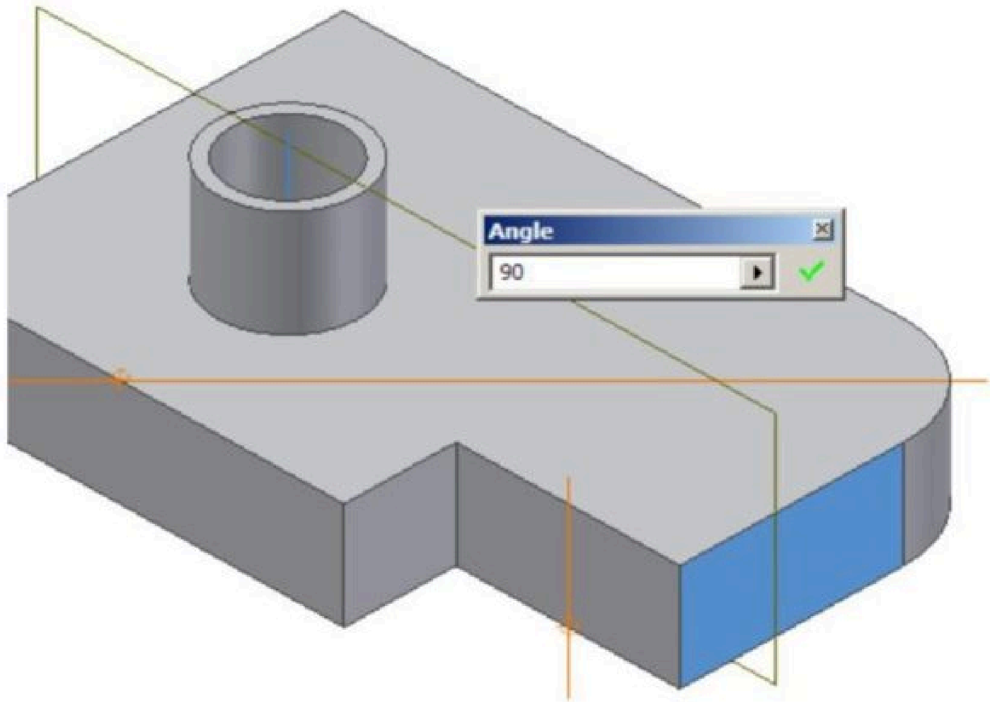


Figure Step 13C

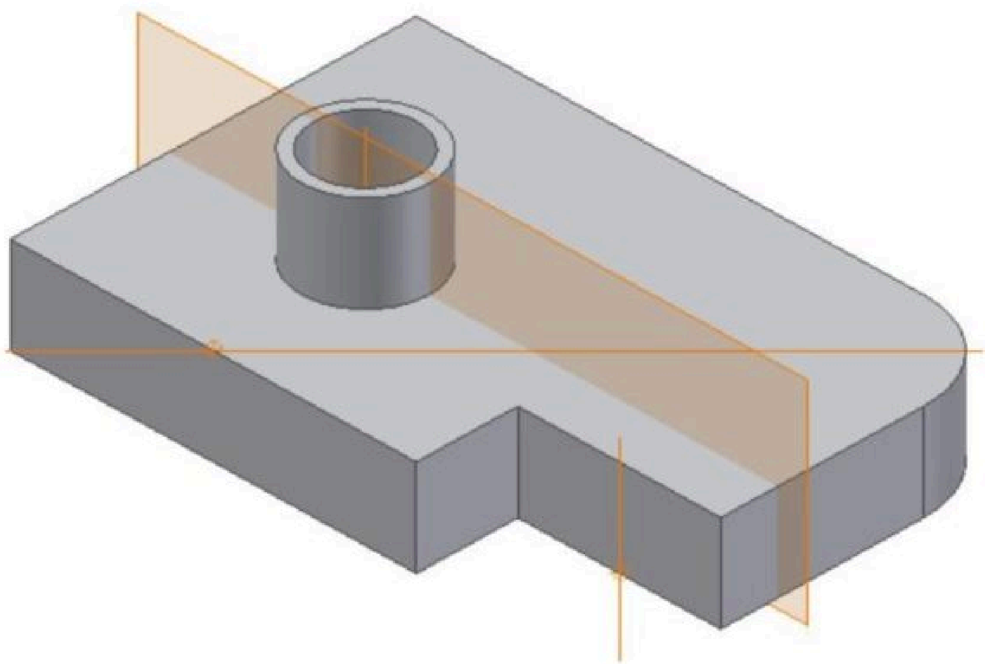


Figure Step 13D

Step 14

Insert an offset Work Plane. Enter the WORK PLANE command and then Offset from Plane. Select the face on the right side. Hold down the left mouse button and drag the plane into the model. The Offset dialogue box will display. Enter the offset distance of -1.25 inches. (Figure Step 14A, 14B, and 14C)

> **AUTHOR'S COMMENTS:** The offset is negative since that is the negative Z direction of the work plane.

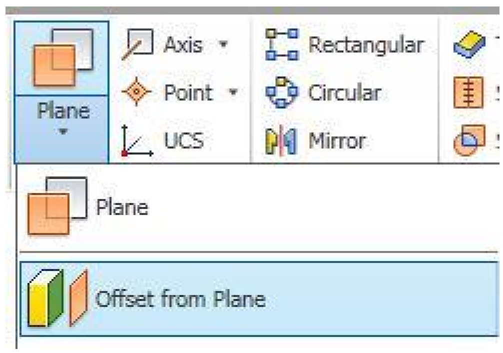


Figure Step 14A

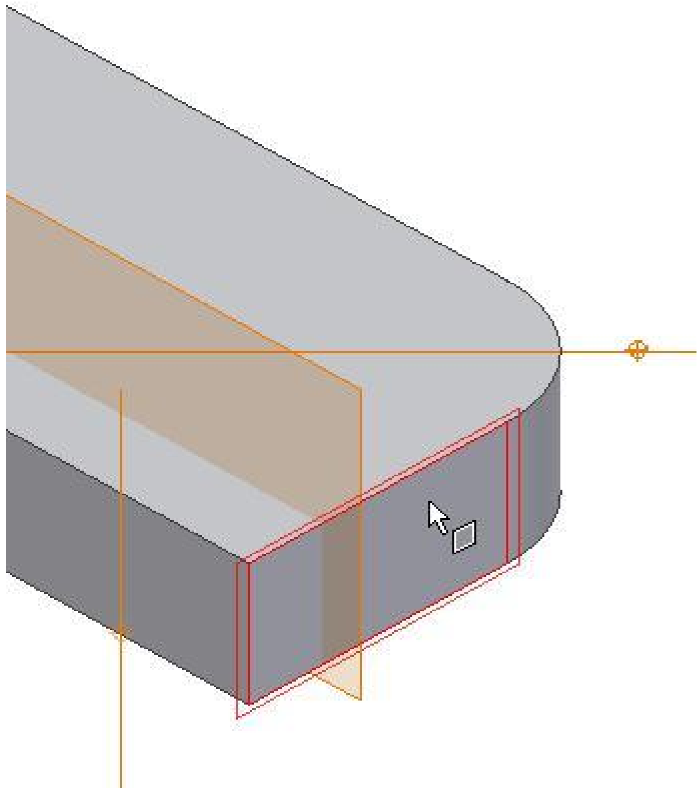


Figure Step 14B

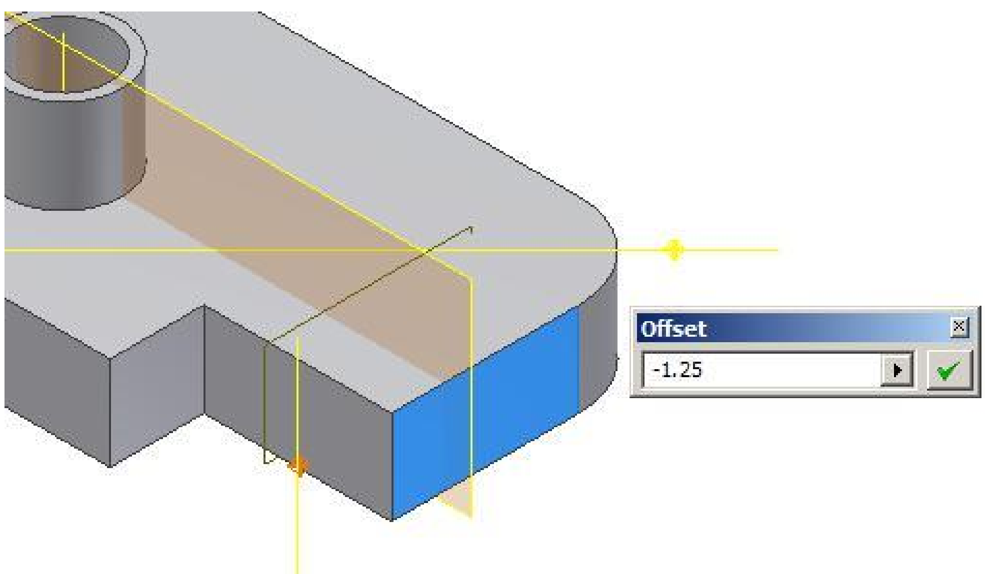


Figure Step 14C

Step 15

Move the cursor to the edge of the newly inserted work plane and when the double arrow icon appears, press and hold down the left mouse button. While holding it down, drag it to enlarge the plane. Do this for all four corners. (Figure Step 15A and 15B)

AUTHOR'S COMMENTS: To get the double arrow to appear, you sometimes have to move the cursor to the corner and click the mouse.

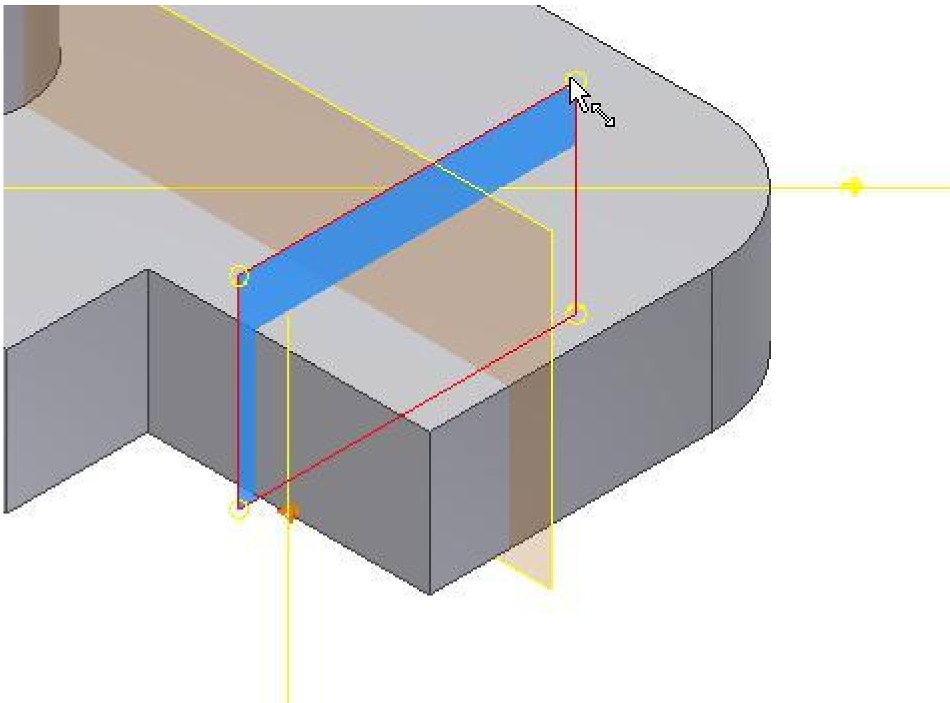


Figure Step 15A

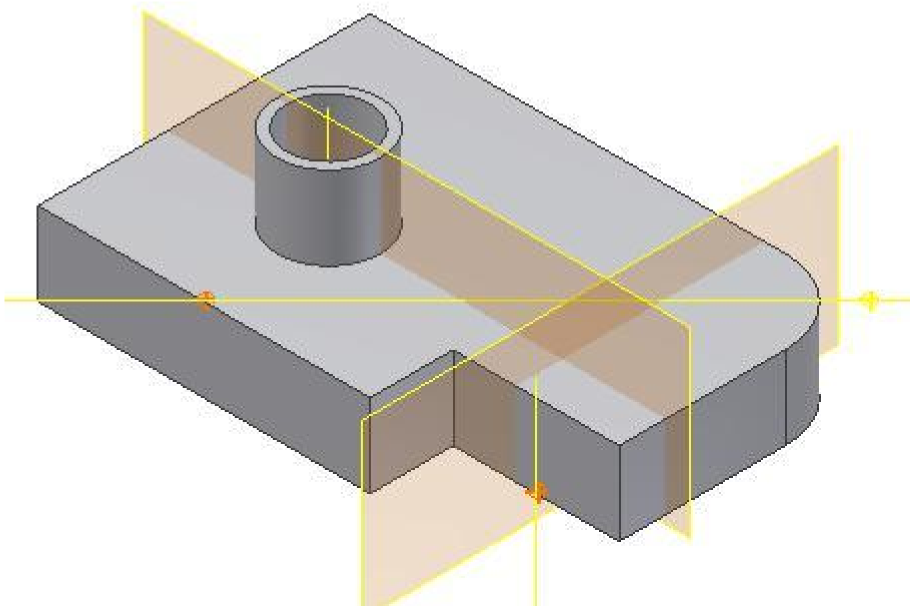
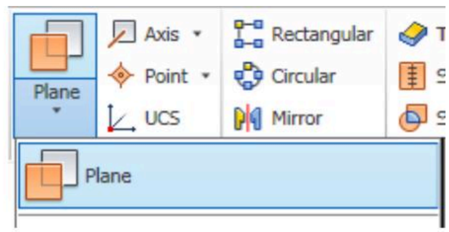


Figure Step 15B

AUTHOR'S COMMENTS: Work axes and work planes can be expanded to any size you require.

Step 16

Insert a Work Plane on a work axis and at an angle to an existing plane. Enter the WORK PLANE command and then Plane. Select the Work Axis shown in the figure. Select the bottom plane next and enter the angle of 90 degrees. (Figure Step 16A, 16B, 16C, 16D, and 16E)



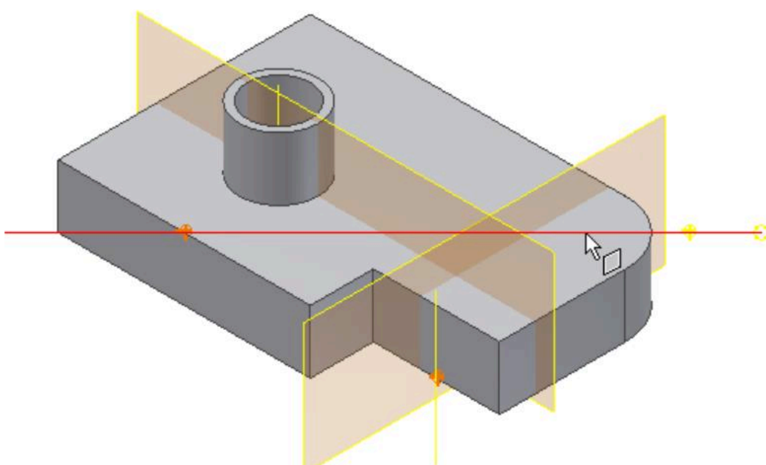


Figure Step 16A

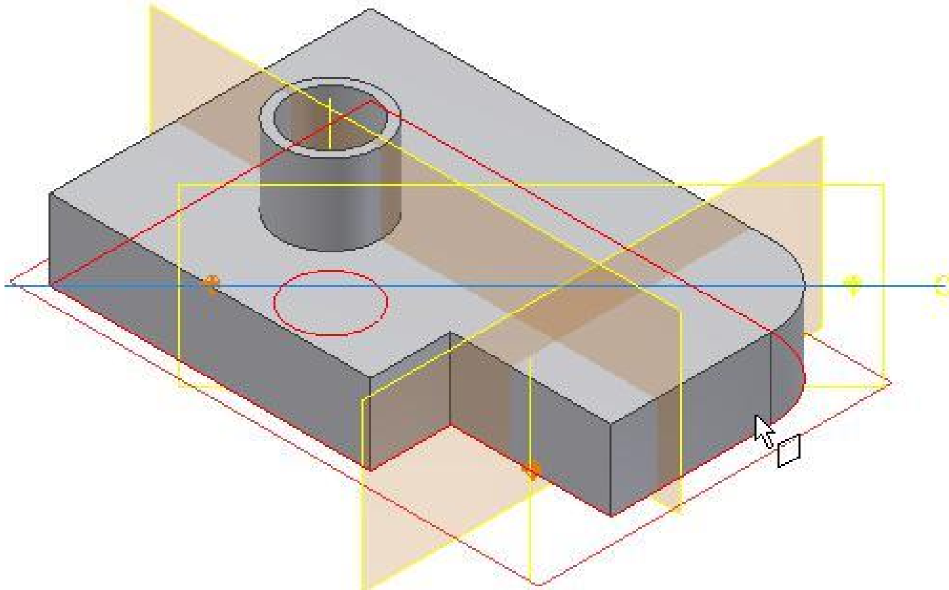


Figure Step 16B

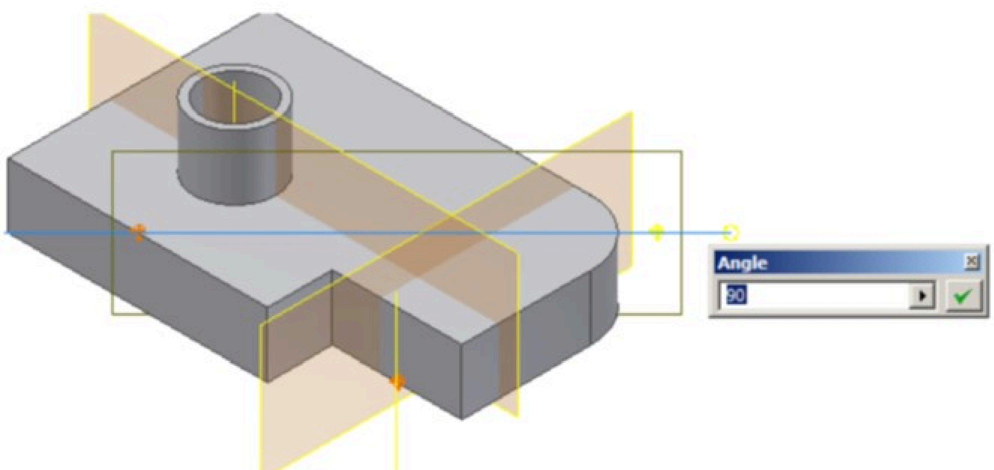


Figure Step 16D

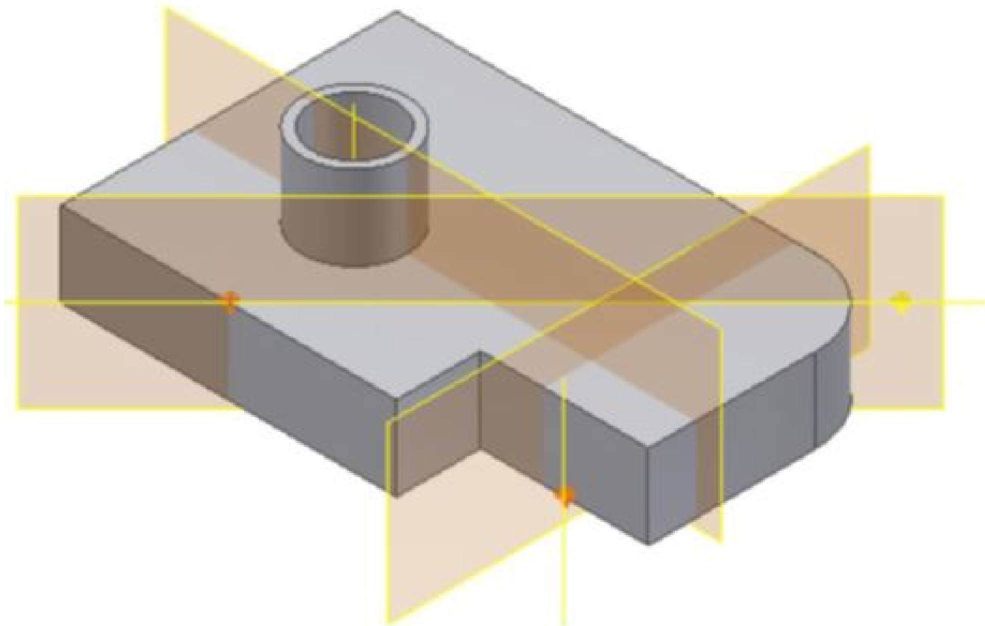


Figure Step 16E

Step 17

The Browser bar should now appear as shown in the figure. (Figure Step 17)

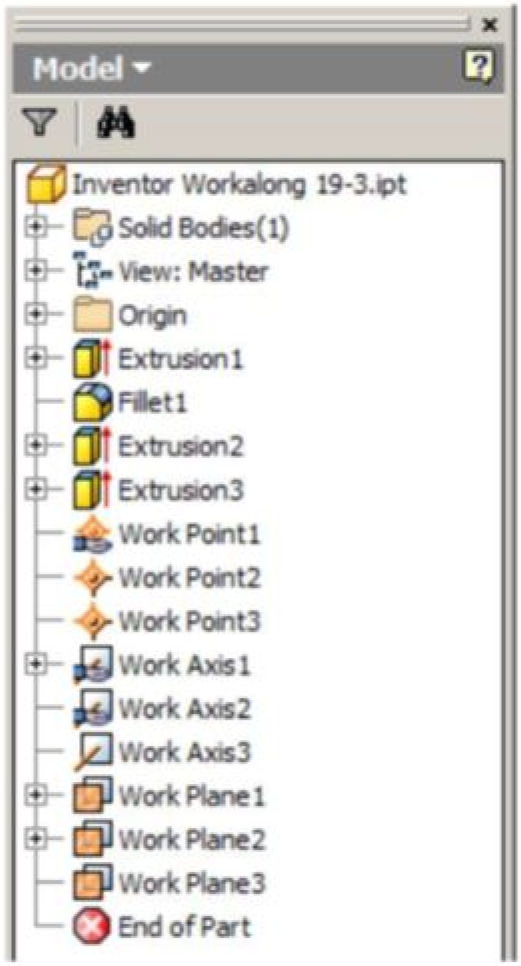


Figure Step 17

Step 18

Disable the visibility of all Work Features. Your solid model should appear as shown in the figure. (Figure Step 18)

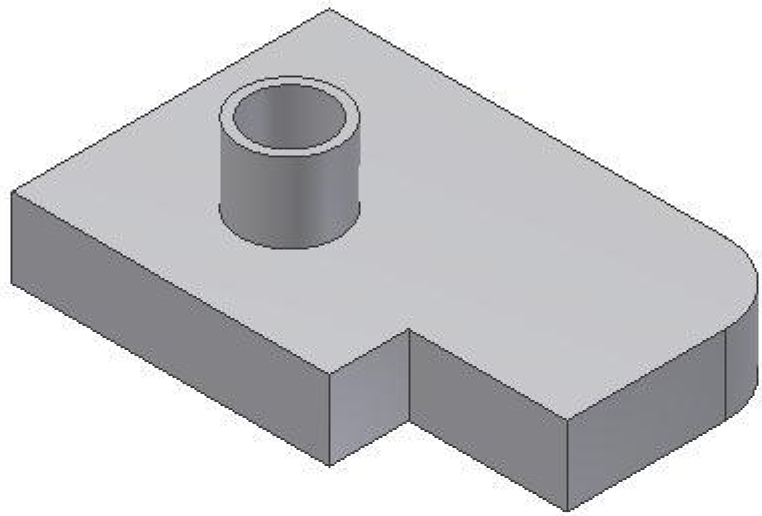


Figure Step 18

AUTHOR'S COMMENTS: You will now be using the drawings shown in Figures 19-6 and 19-7 and inserting Work Features to add the detail on the model.

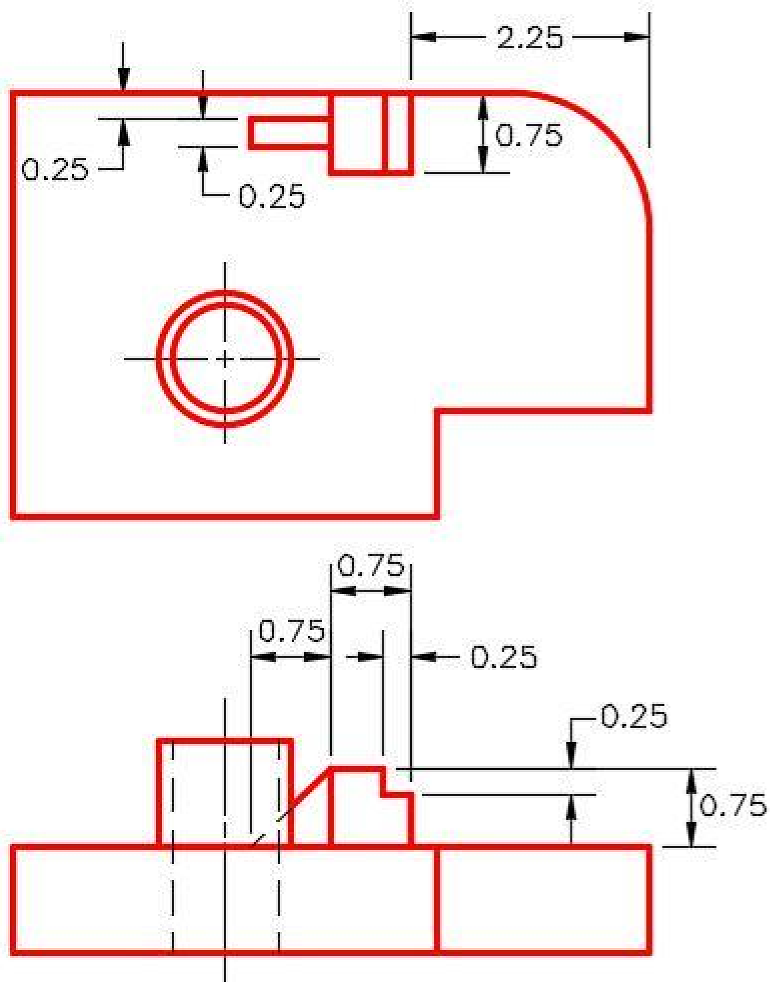


Figure 19-6
Dimensioned Multiview Drawing

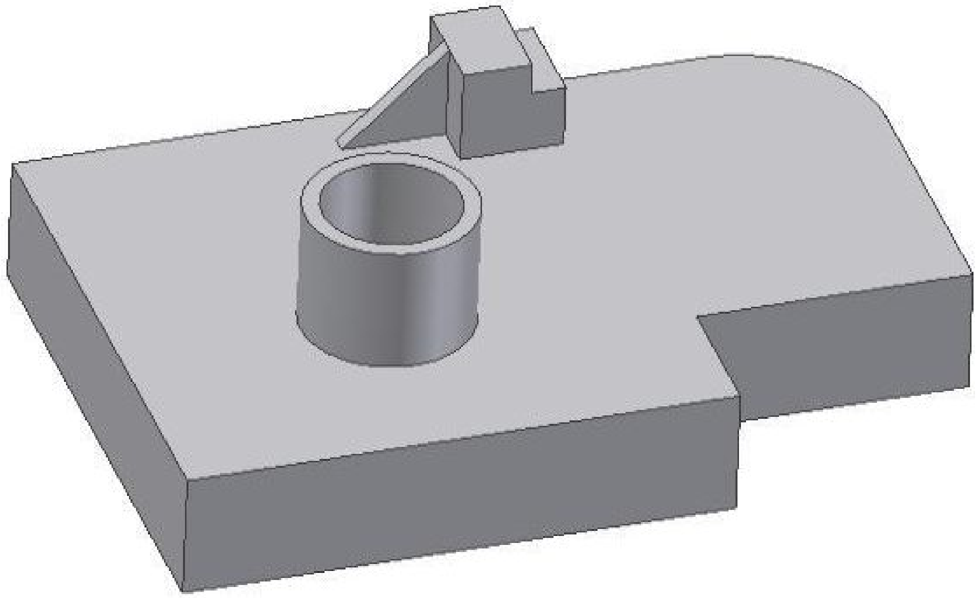


Figure 19-7
3D Model – Home View

Step 19

Using the offset method, insert a Work Plane -2.25 inches from the right side face. Expand the size of the Work Plane. Start a new sketch on the Work Plane. (Figure Step 19A and 19B)

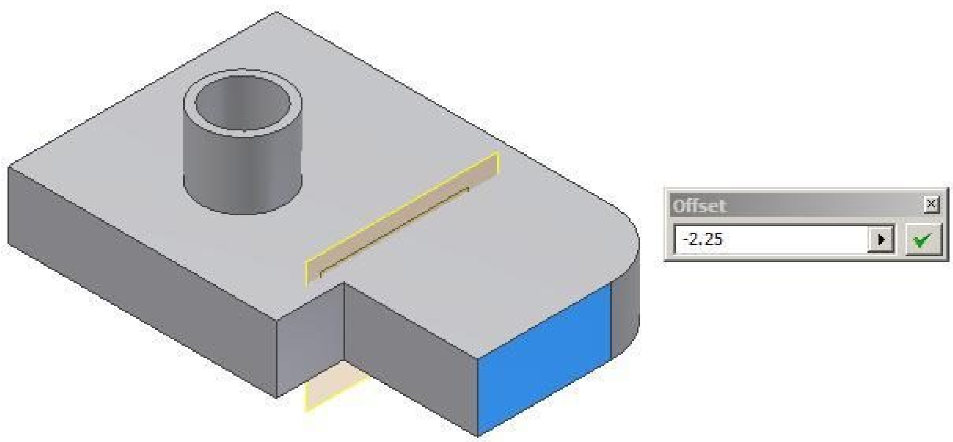


Figure Step 19A

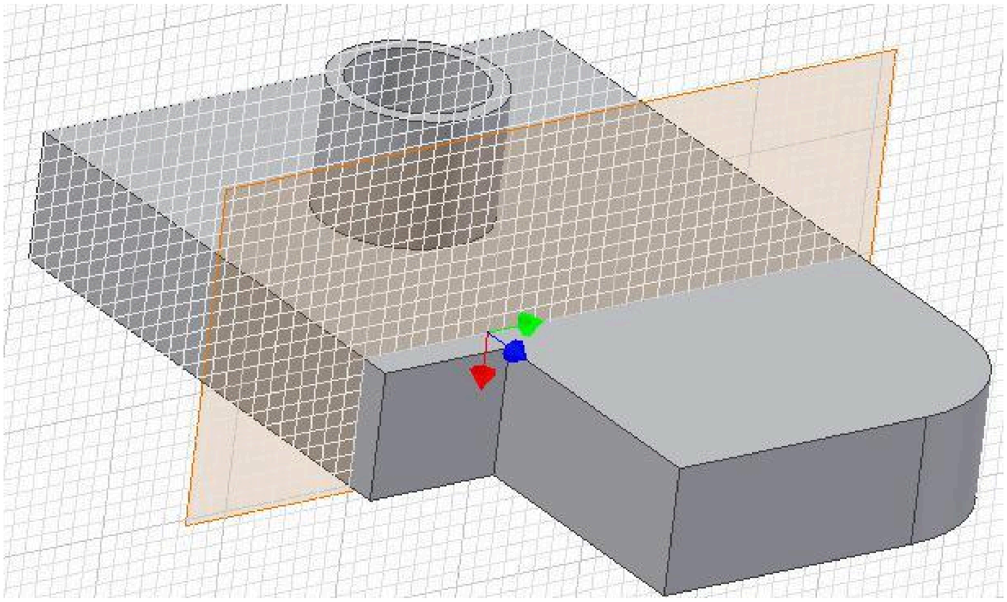


Figure Step 19B

Step 20

Enter the TWO POINT RECTANGLE command and right click the mouse. Ensure that AutoProject is enabled. Draw a rectangle by snapping to the bottom right corner. Dimension the square and ensure it is fully constrained. Extrude the sketch. (Figure Step 20A, 20B, 20C, and 20D)

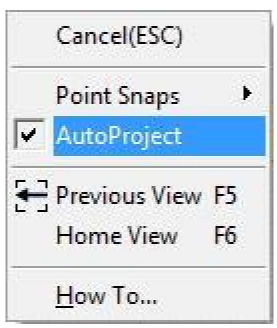


Figure Step 20A

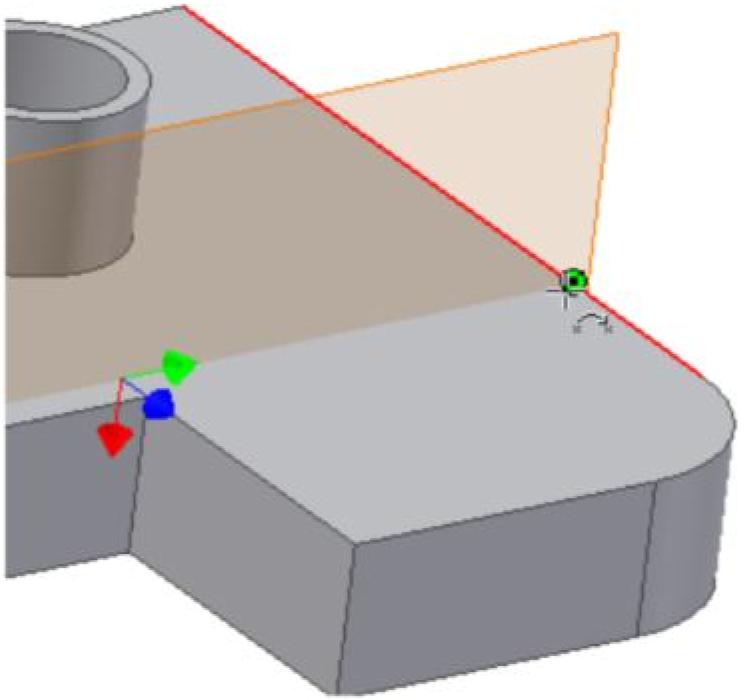


Figure Step 20B

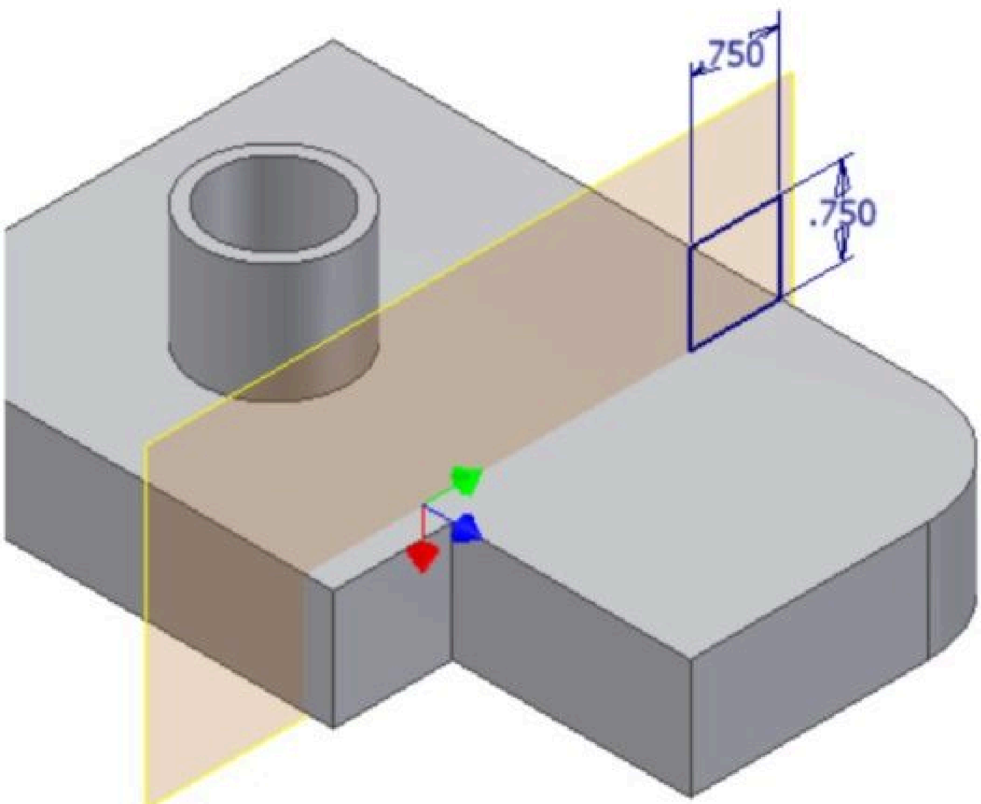


Figure Step 20C

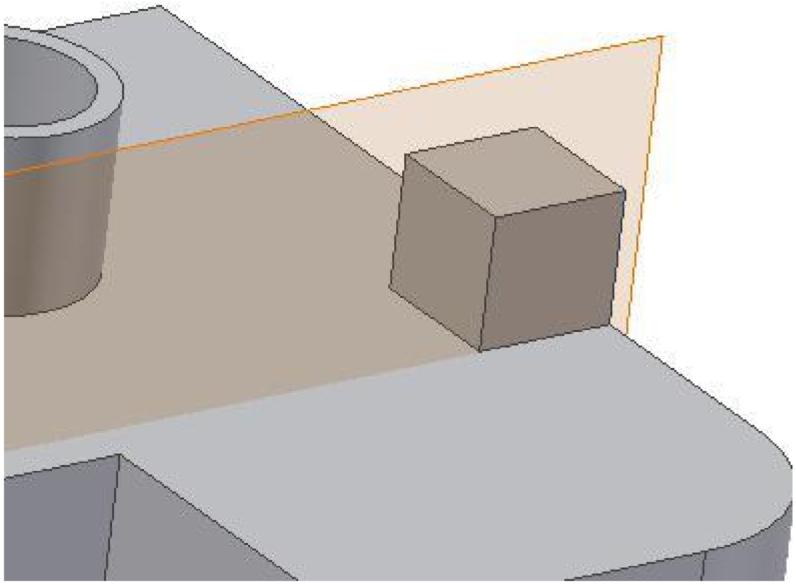


Figure Step 20D

Step 21

Start a new sketch and draw two lines. Dimension and extrude it. (Figure Step 21A and 21B)

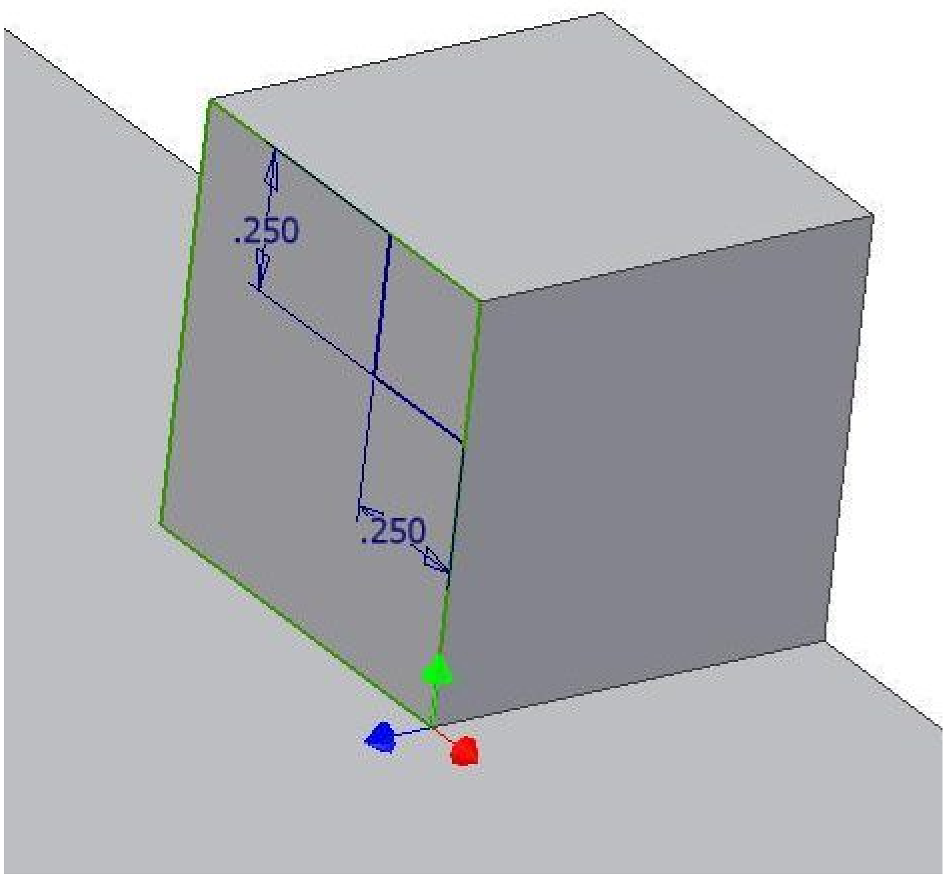


Figure Step 21A

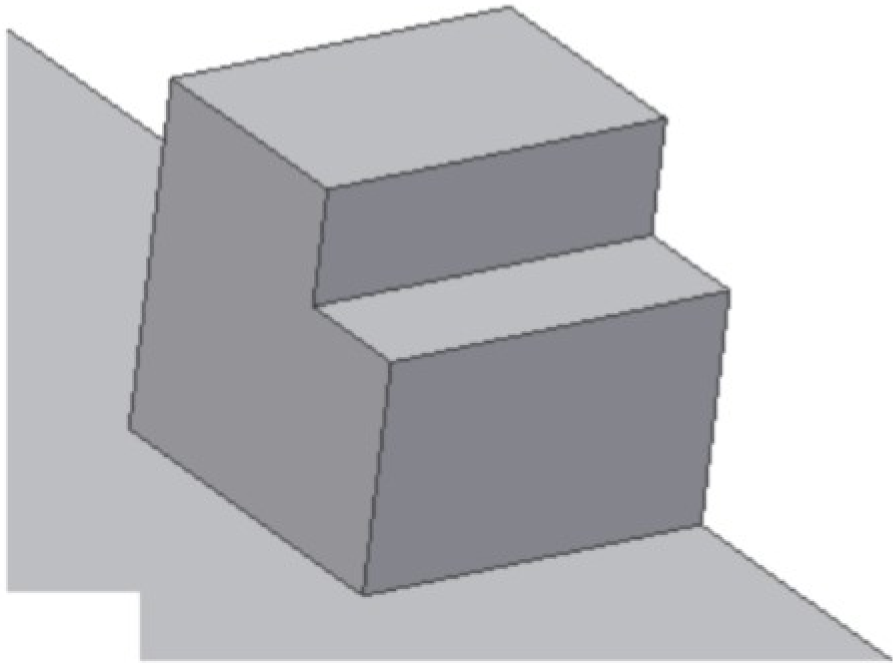


Figure Step 21B

Step 22

Insert a Work Plane -0.375 inches in from the front face. (Figure Step 22)

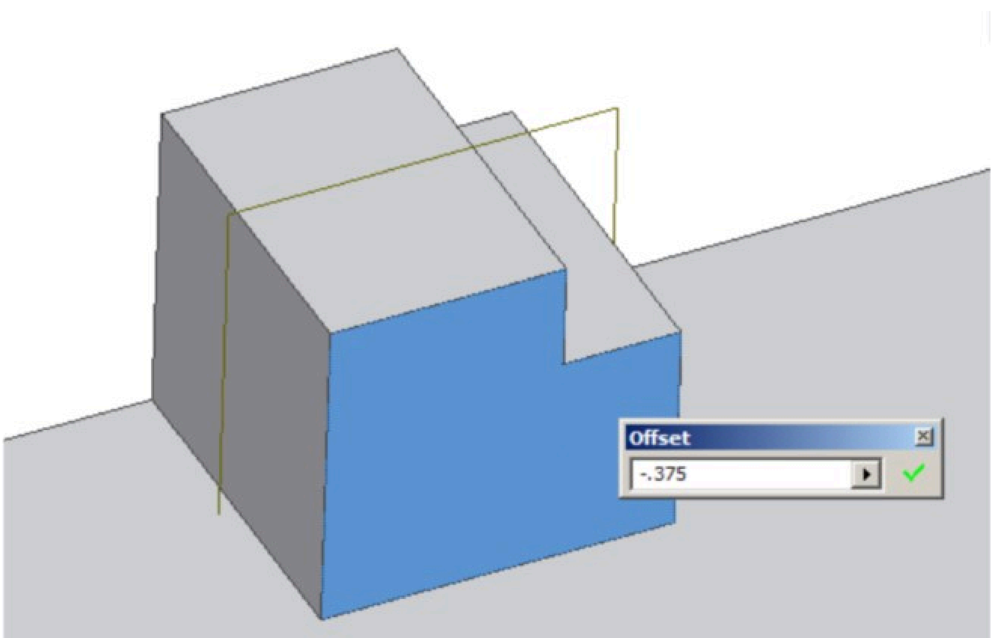


Figure Step 22

Step 23

Draw a triangle and insert two dimensions to fully constrain it. (Figure Step 23)

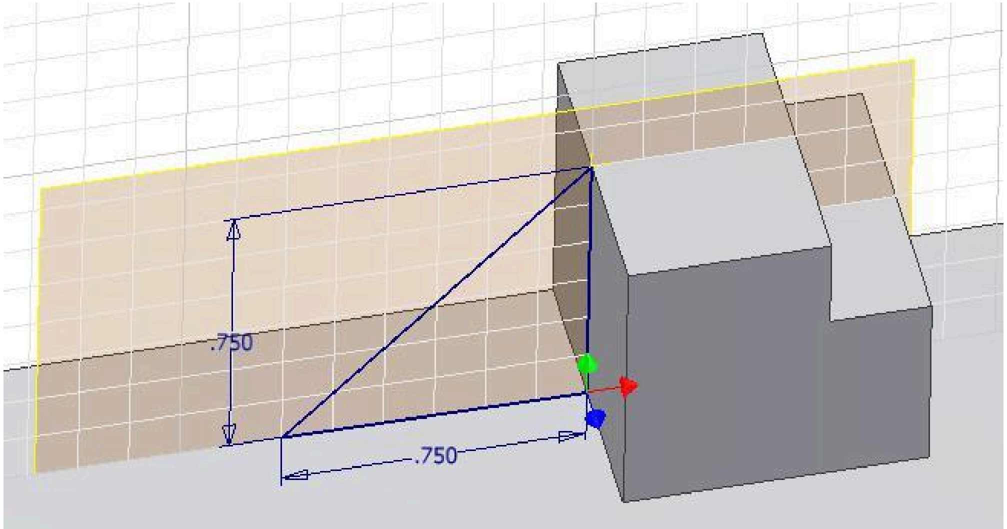


Figure Step 23 [Click to see image full size]

Step 24

Extrude the sketch 0.125 inches in both directions to complete the part. (Figure Step 24)

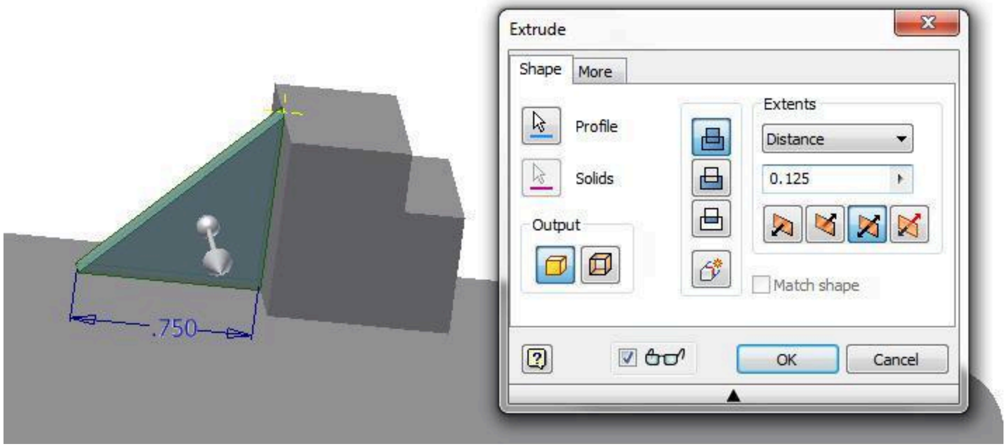


Figure Step 24 [Click to see image full size]

AUTHOR'S COMMENTS: I placed the sketch at the centre of the rib so I extruded it one-half its width in both directions.

Step 25

Disable the visibility of all Work Features. The completed solid model should appear as shown in the figure. (Figure Step 25)

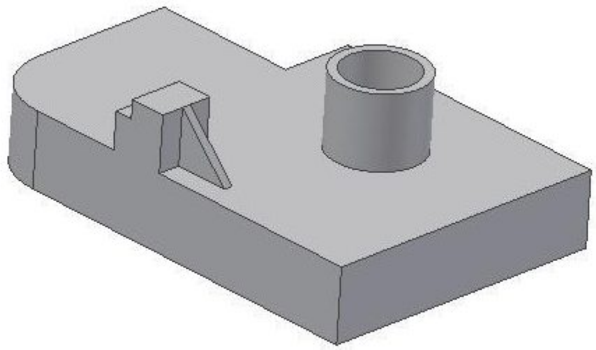


Figure Step 25

AUTHOR'S COMMENTS: As you can see there are may ways to insert and use Work Features. Practice inserting them as you construct future models. Read Inventor's Help files if you have trouble.

Step 26

Change to the Home view. (Figure Step 26)

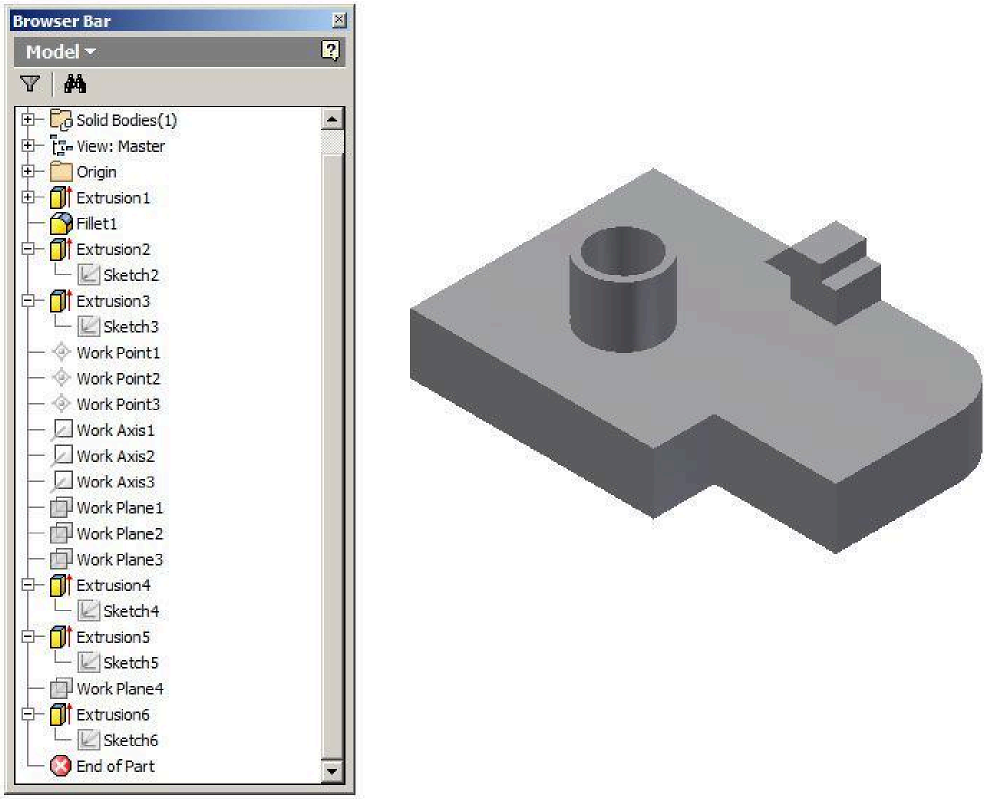


Figure Step 26 [Click to see image full size]

Step 27

Save and close the part.

Key Principles

Key Principles in Module 19

1. The POLYGON command is used to draw a regular polygon on a 2D sketch. You can select the number of sides and choose between either an inscribed or a circumscribed polygon.
2. The threads created by the THREAD command are not actual threads constructed on the model. They are simply a graphical representation of the threads.
3. The TANGENT CIRCLE command is used to draw a circle tangent to three lines.
4. The AUTO DIMENSION command is used to add dimensions or constrains automatically to fully constrain a sketch.
5. The three Work Features available in Inventor are the Work Point, the Work Axis, and the Work Plane.

6. A Work Point is a parametric construction point or a single XYZ location that is inserted and then used as a Work Feature.
7. A Work Axis is a parametric construction line or two XYZ locations joined by a line that is inserted and then used as a Work Feature.
8. A Work Plane is parametric construction plane or four XYZ locations joined by lines inserted on the model or Model space and then used as a Work Feature.

Lab Exercise 19-1

Time allowed: 90 minutes.

Part Name	**Project**	**Units**	**Template**	**Color**	**Material**
Inventor Lab Lab 19-1	Inventor Course	Millimeters	Metric – Modules Part (mm).ipt	Chrome – Black Polished	N/A

Step 1

Draw the Base sketch on the Front view.

Step 2

Project the Center Point onto the Base plane.

Step 3

Note the location of X0Y0Z0. Draw the necessary sketches and EXTRUDE them to produce the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to maintain the objects shape and size. (Figure Step 3A, 3B, 3C, 3D, and 3E)

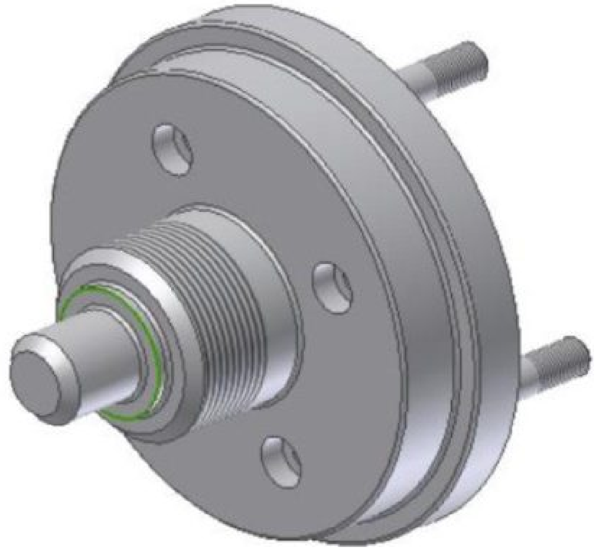


Figure Step 3A
3D Model – Orbited View

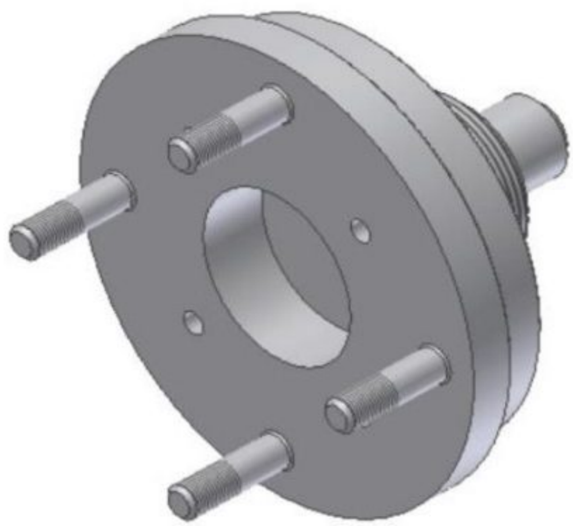


Figure Step 3B
3D Model – Home View

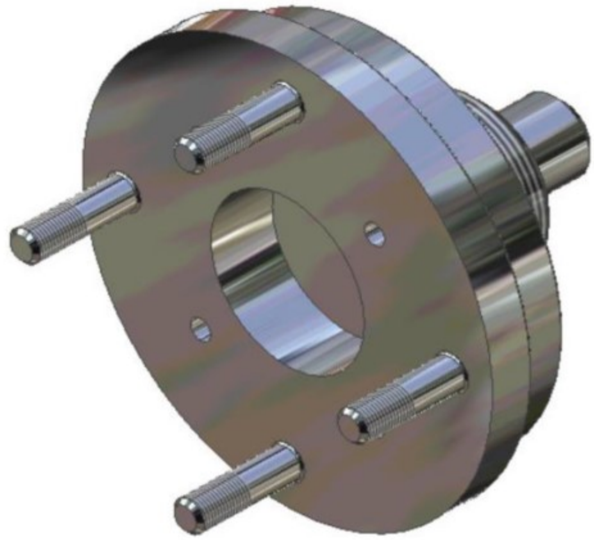


Figure Step 3C
Solid Model – Home View

Figure Step 3D
3D Model – Orbited View

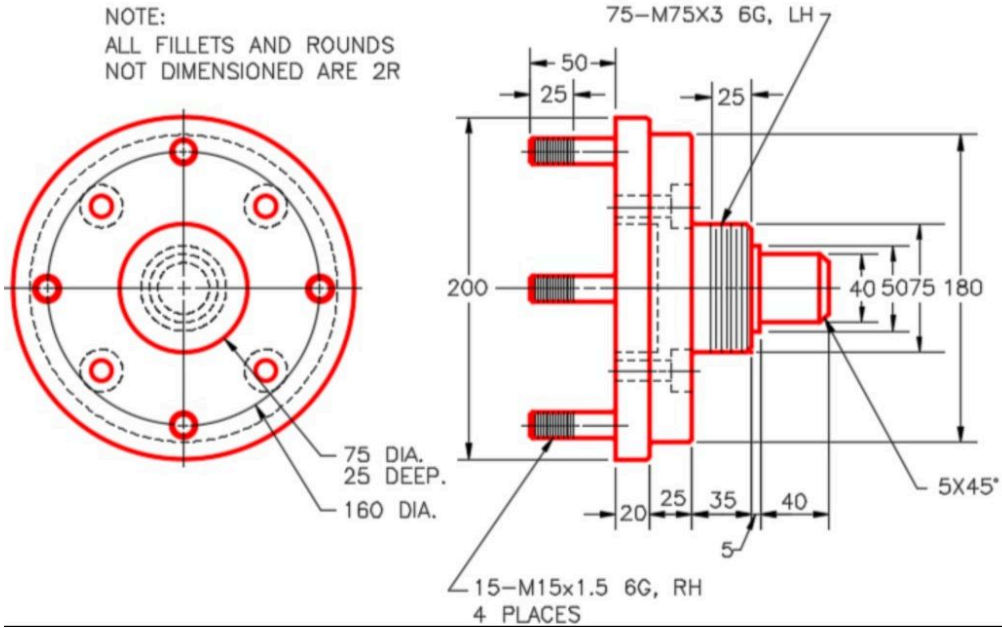


Figure Step 1E
Left Side View – Multiview Drawing [Click to see image full size]

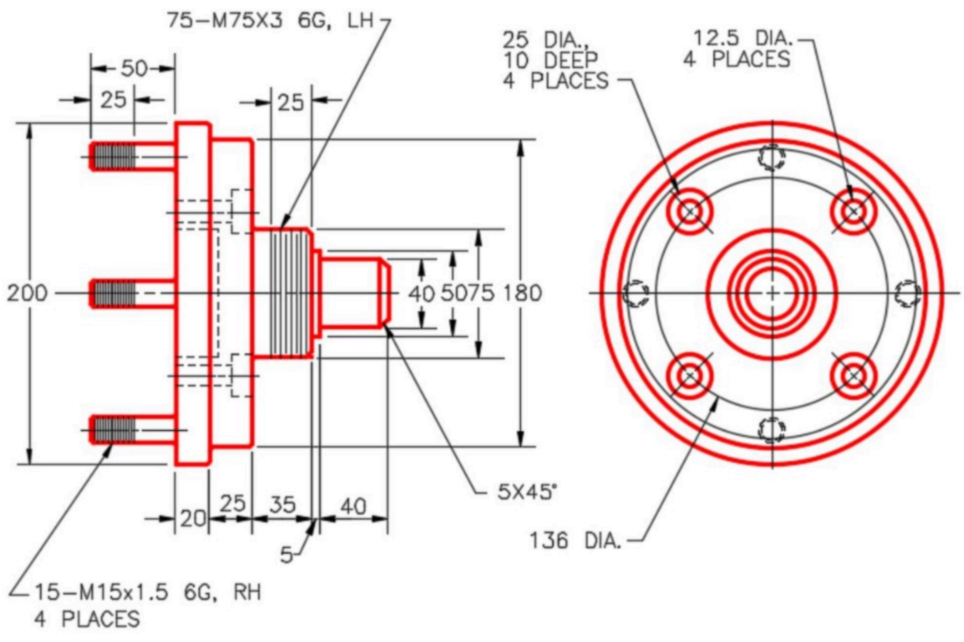


Figure Step 1F
Right Side View – Multiview Drawing [Click to see image full size]

Step 4

Create the fillets and chamfers after the model is totally constructed.

Step 5

Apply the colour shown above.

Lab Exercise 19-2

Time allowed: 90 minutes.

Part Name	Project	Units	Template	Color	Material
Inventor Lab Lab 19-2	Inventor Course	Millimeters	Metric – Modules Part (mm).ipt	Chrome – Black Polished	N/A

Step 1

Draw the Base sketch on the Right Side view.

Step 2

Project the Center Point onto the Base plane.

Step 3

Note the location of X0Y0Z0. Draw the same model you just drew in Lab Exercise 19-1. In this exercise, REVOLVE the Base sketch and then add sketches and extrude them to complete the model. Apply all of the necessary geometrical and dimensional constraints to maintain the objects shape and size. (Figure Step 3A, 3B, 3C, and 3D)

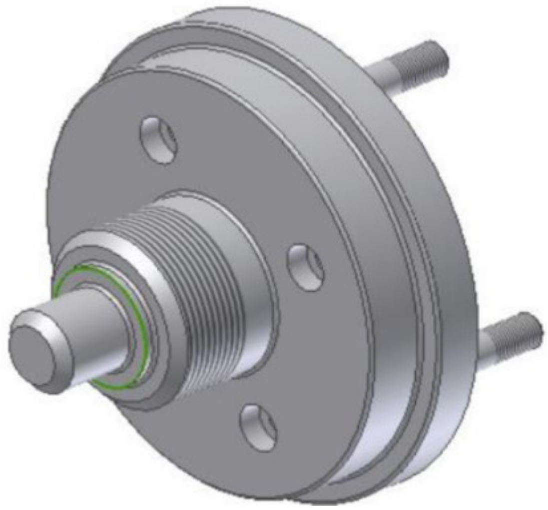


Figure Step 3A
3D Model – Orbited View

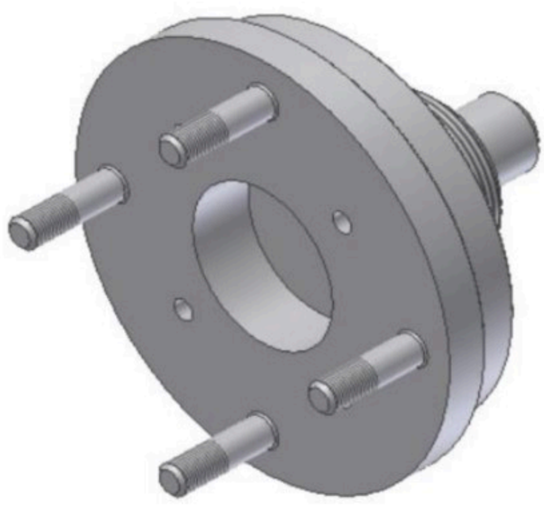


Figure Step 3B
3D Model – Home View

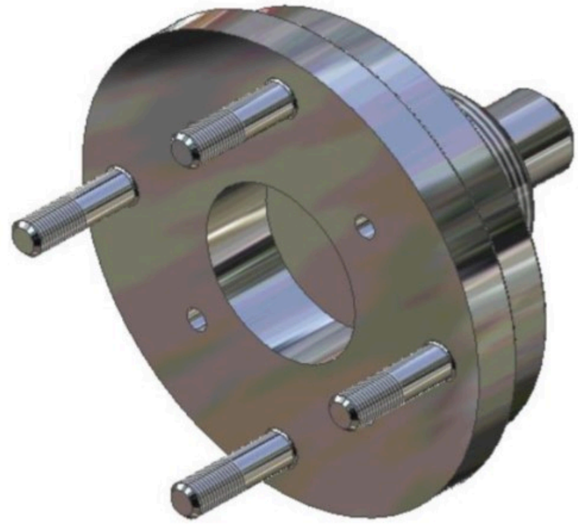
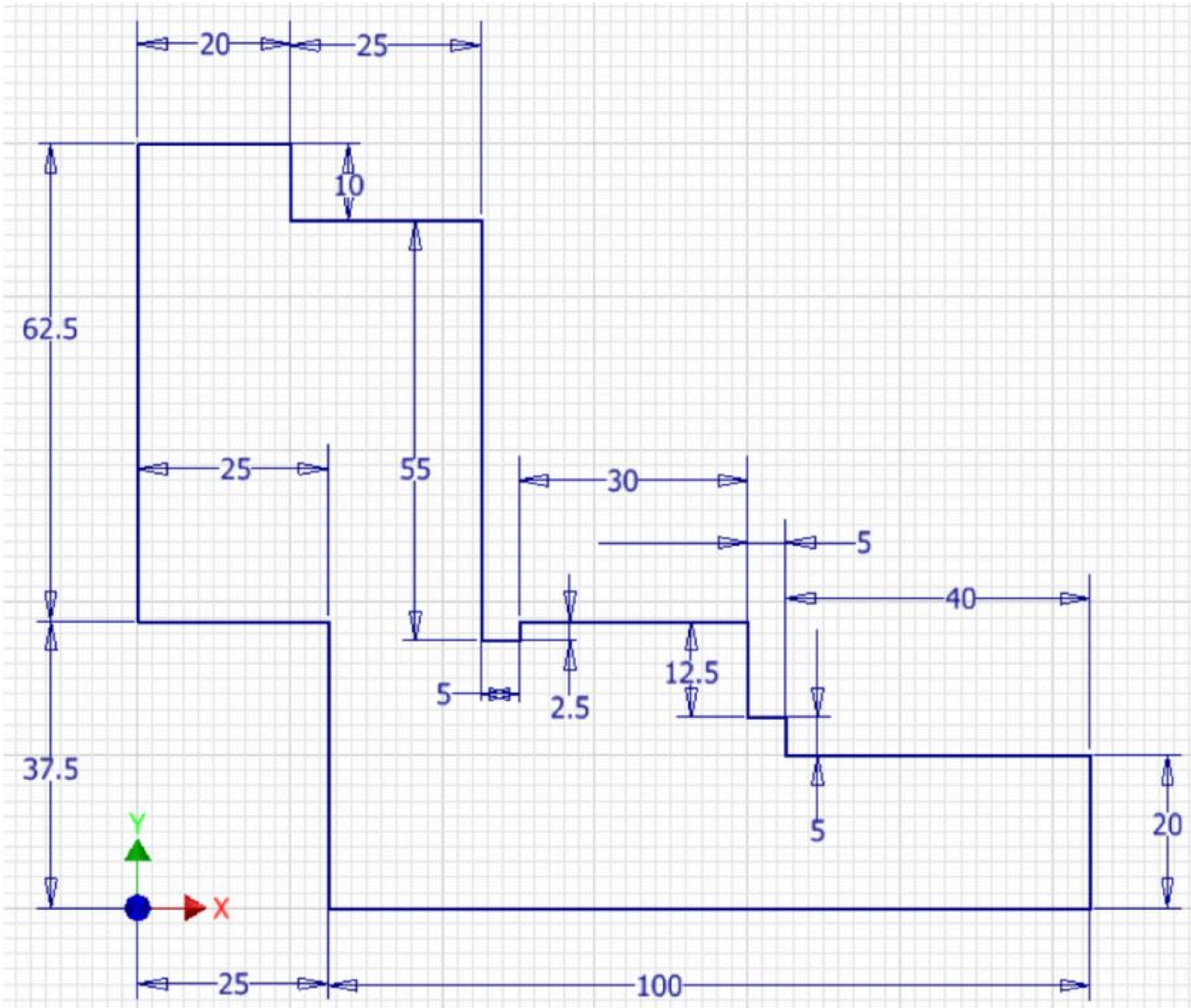
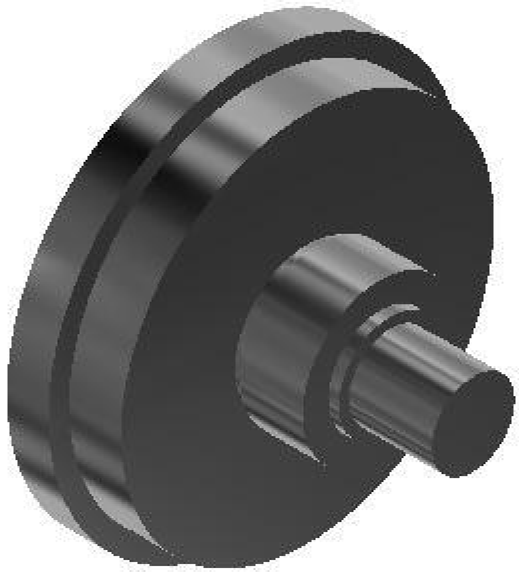


Figure Step 3C
Solid Model – Home View

Figure Step 3D
Solid Model – Orbited View



Author's Base Sketch



Author's Base Model

Step 4

Create the fillets and chamfers after the model is totally constructed.

Step 5

Apply the colour shown above.

Lab Exercise 19-3

Time allowed: 90 minutes.

Part Name: Inventor Lab 19-3	**Project**: Inventor Course	**Units**: Millimeters
Template: Metric – Modules Part (mm).ipt	**Color**: Galvanized (texture)	**Material**: N/A

Step 1

Project the Center Point onto the Base sketch.

Step 2

Draw the necessary sketches and extrude or revolve them to produce the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to maintain the objects shape and size. (Figure Step 2A, 2B, 2C, 2D, and 2E)

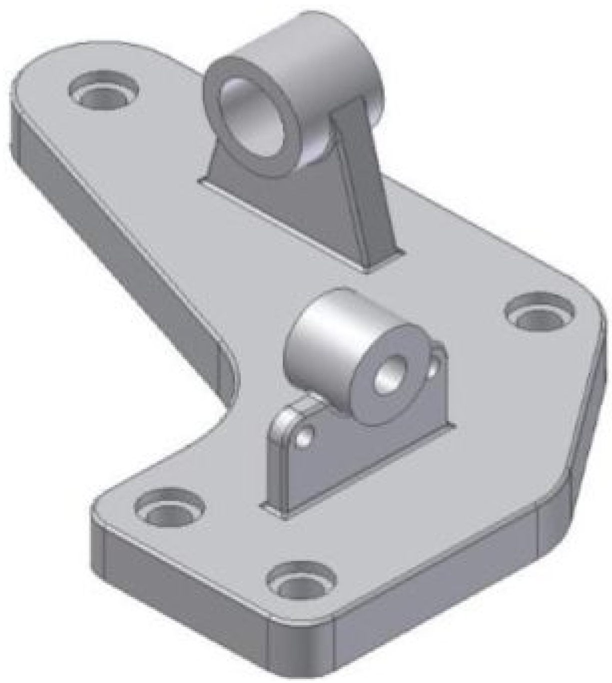


Figure Step 2A
3D Model – Home View

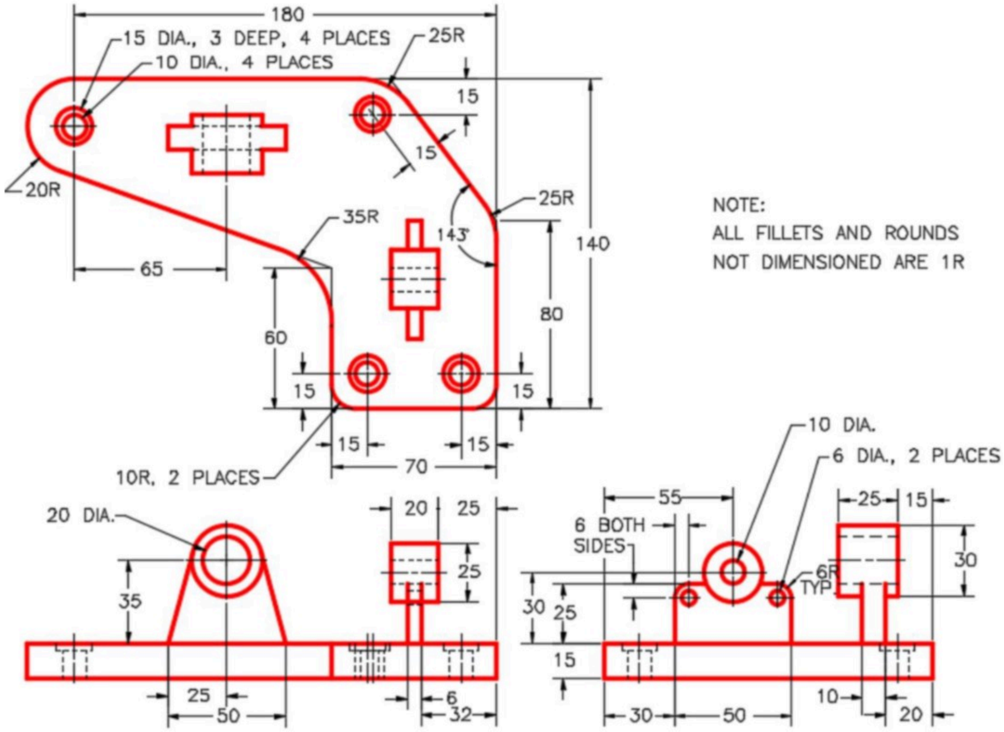


Figure Step 2B [Click to see image full size]

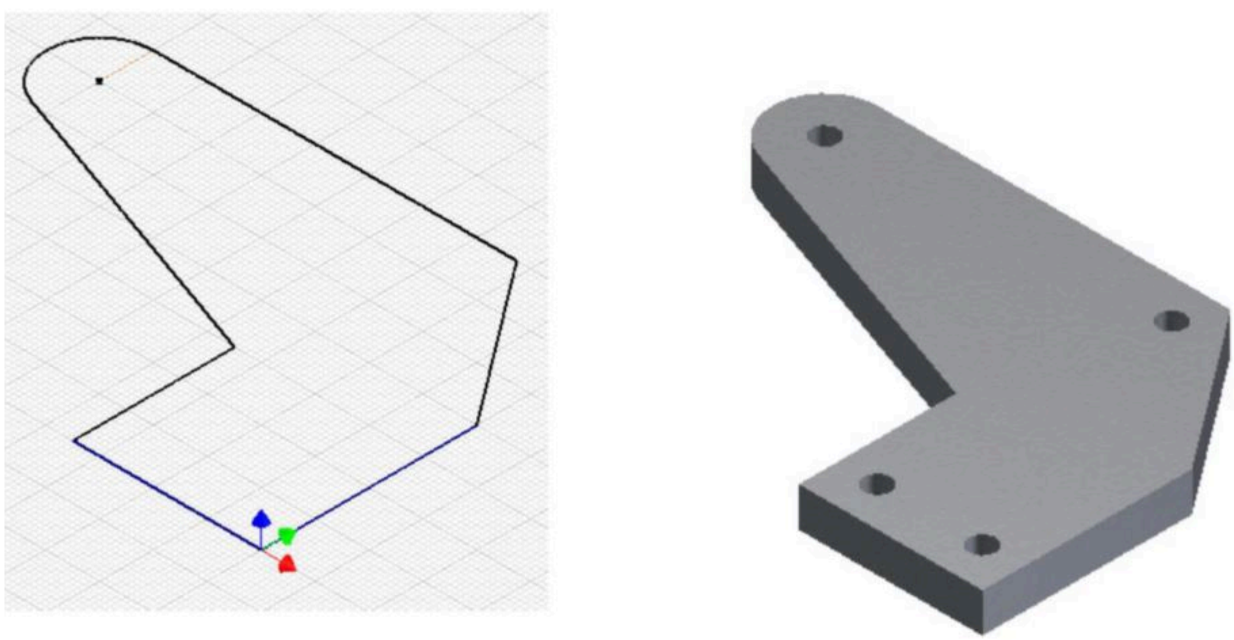
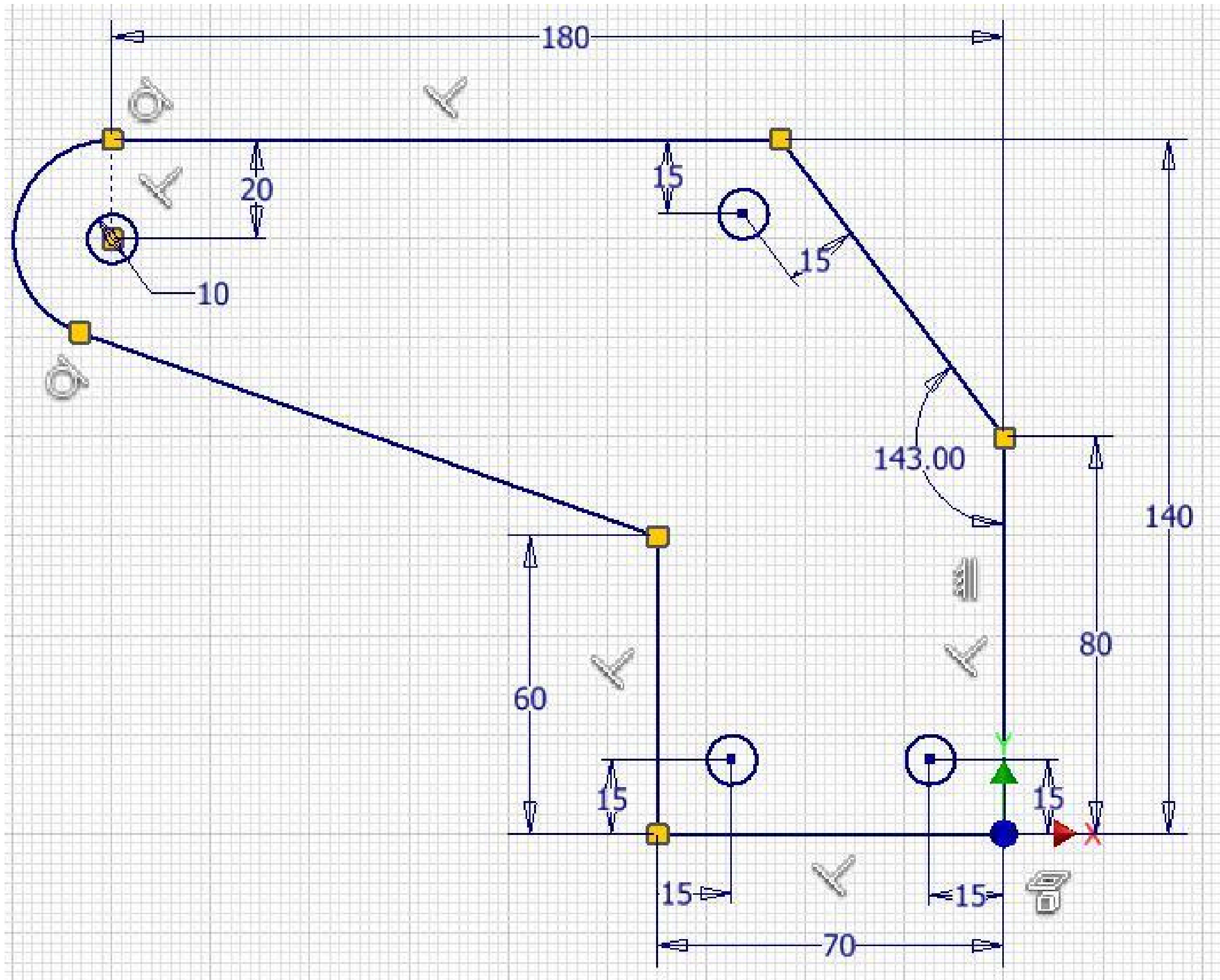


Figure Step 2C
Suggested Base Sketch – Author's Base Model Top (XY) Plane



Author's Base Sketch

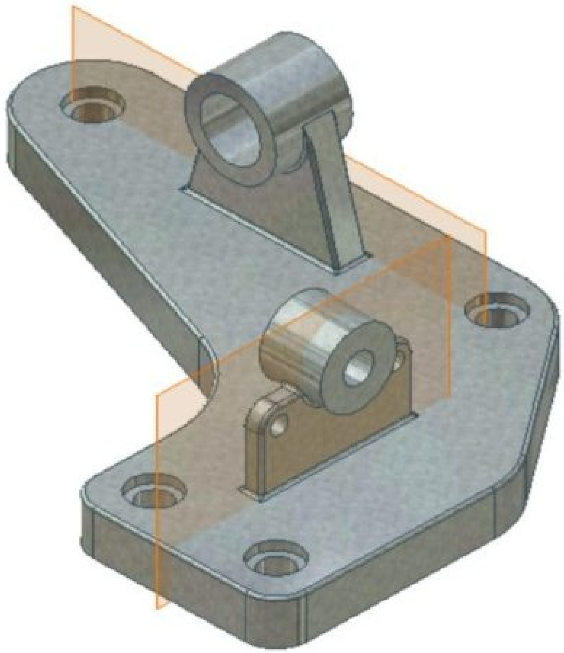


Figure Step 2D
Solid Model – Home View
With Work Planes

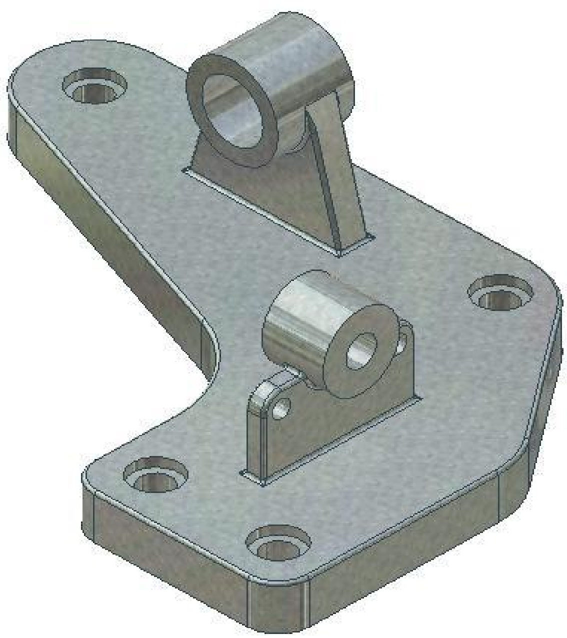


Figure Step 2E
Solid Model – Home View

Step 3

Create the fillets and chamfers after the model is totally constructed.

Step 4

Apply the colour shown above.

Module 20 Modifying Solid Models

Learning Outcomes

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

1. Describe how to edit the dimensions and part features of an existing solid model using the Browser bar to aid you.
2. Describe how to hide and/or suppress features in the solid model.
3. Describe and apply the MEASURE command to measure lengths, loops, angles, or areas of a solid model in either two dimensions or three dimensions.
4. Describe how to set the material and change the colour of faces of the solid model.
5. Describe how to obtain the physical properties of a solid model.

Modifying Solid Models

The ability to modify solid models is as important to the drafter/designer as being able to construct models. Since most parts are modified after the initial design or revised and used in another project, it is very important that you can modify solid models rather then redraw them. The true power of Inventor is its ability to modify a solid and have it conform to the geometrical and dimensional constraints applied to it when it was constructed without having to redraw it. In this module, the basics of modifying solid models is taught. The Inventor Advanced book will cover the more advanced methods of modifying solid models.

Working with the Browser Bar

The Browser bar is used extensively as a tool to assist you when modifying solid models. It displays the work features and the part features for the current part.

Features

Work features are the basic sketching planes (XY, XZ, YZ), the axes (X, Y, Z) and the Center Point as shown in Figure 20-1.*Part features* are the 3D features added to the solid model in model construction. The part features for the active part are shown in Figure 20-2.

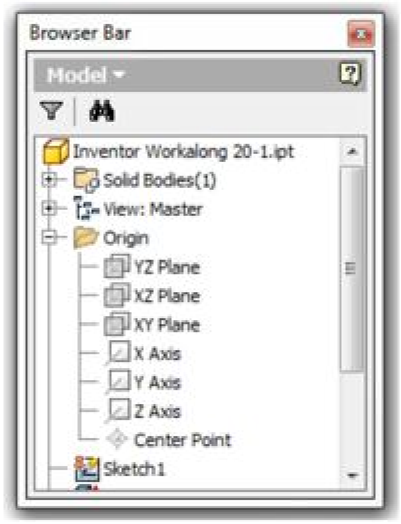


Figure 20-1
Work Features

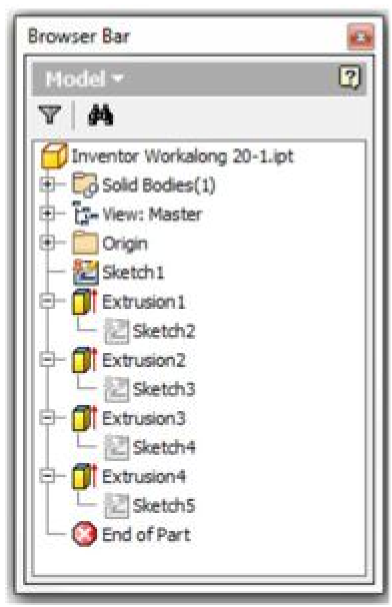


Figure 20-2
Part Features

Edges and Faces

Edges are the lines, circles, or arcs located between the planes that form the solid model. Faces are the planes between the edges of the solid. A face can also be circular or cylindrical in shade, for example the hole in the model. Figure 20-3 shows each visible face of a solid model in a different colour.

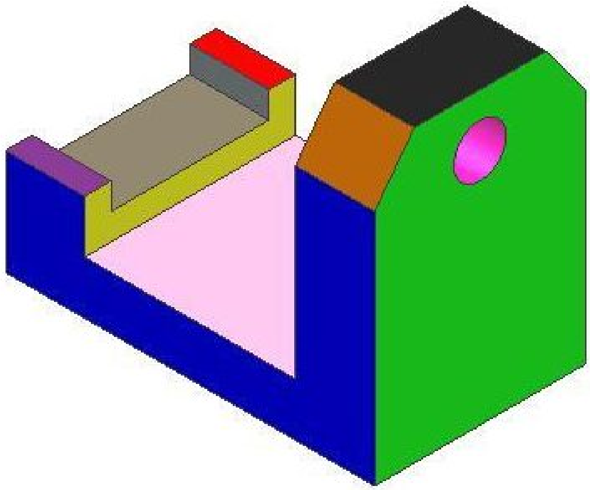


Figure 20-3
Edges and Faces

WORK ALONG: Modifying a Solid Model

Step 1

Open the part file: Inventor Workalong 20-1.ipt. This is one of the parts that you received when you download your book. It should be in the folder: Lab Exercise. (Figure Step 1A and 1B)

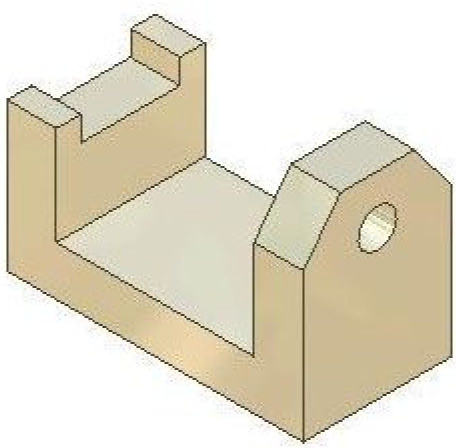


Figure Step 1A

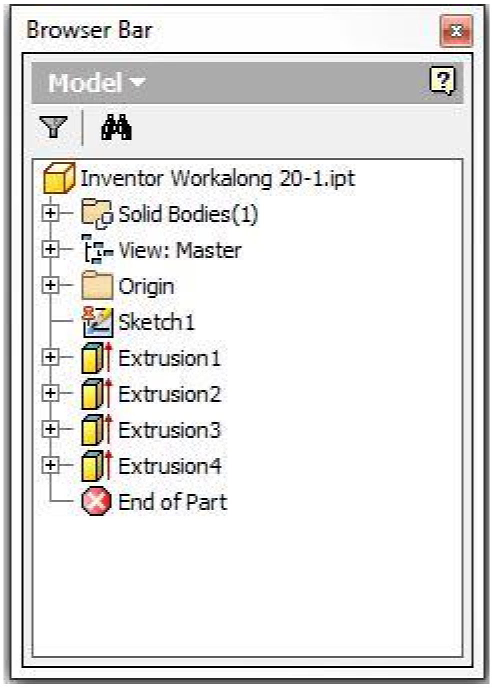


Figure Step 1B

Step 2

The Filter icon located at the top of the Browser bar allows you to enable or disable the visibility of the features in the Browser bar. Click the Filter icon. The filter list will display. (Figure Step 2A and 2B)

Figure Step 2A

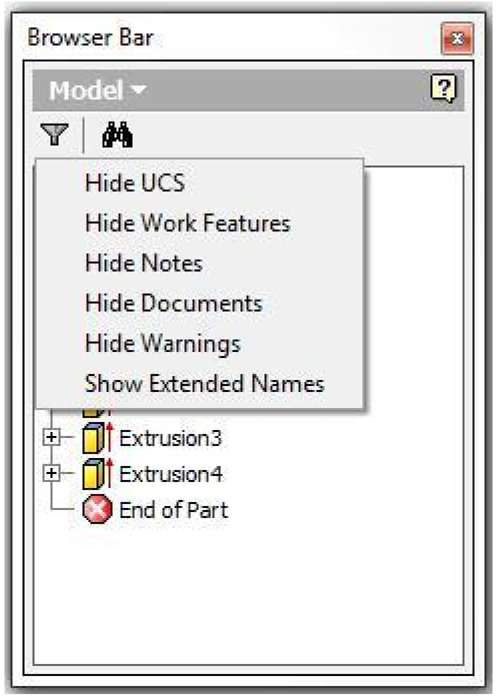


Figure Step 2B

Step 3

Click Hide Work Features to enable it. The work features are now hidden for the current part. (Figure Step 3A and 3B)

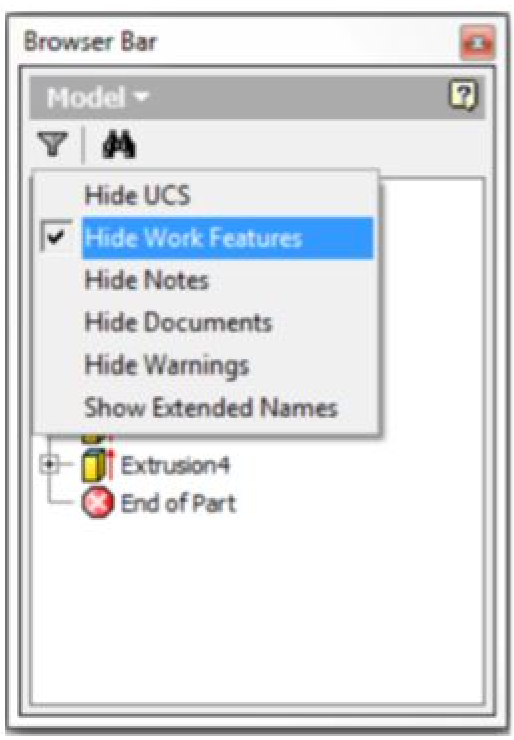


Figure Step 3A

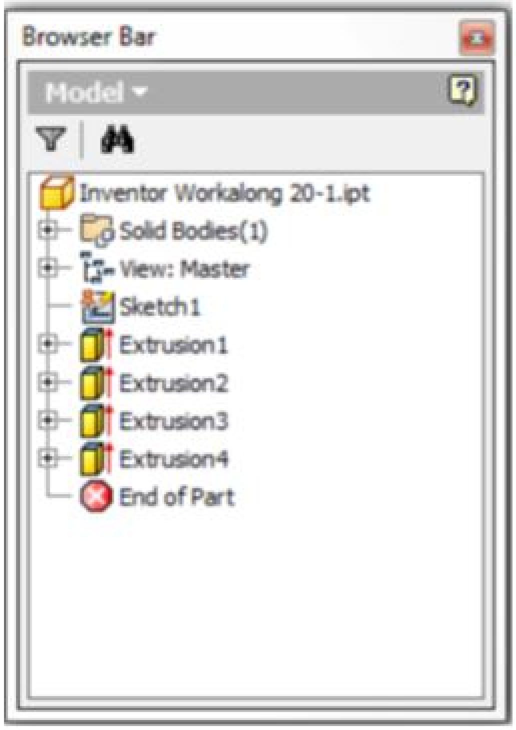


Figure Step 3B

Step 4

Since you want the work features to display most of the time, click the Filter icon again and disable the Hide Work Features. (Figure Step 4)

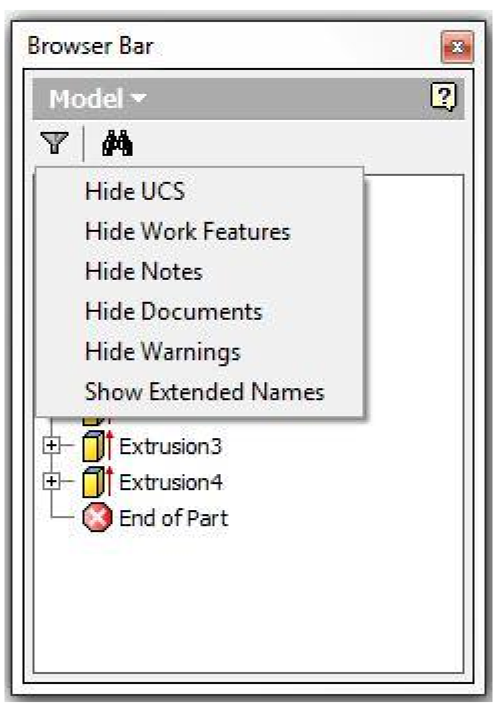


Figure Step 4

Step 5

The Part icon is the first icon inside the Browser bar window. It displays the name of the active part. (Figure Step 5)

Figure Step 5

Step 6

Right-click the part name. In the Right-click menu, click Expand All Children. Note how all the folders and part features will expand to display every feature in the current part. Right click the menu again and this time, click Collapse All Children. (Figure Step 6A, 6B, and 6C)

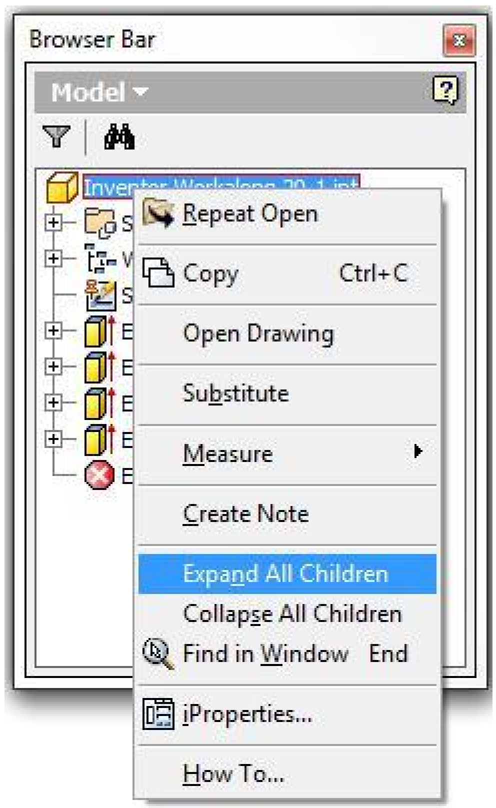


Figure Step 6A

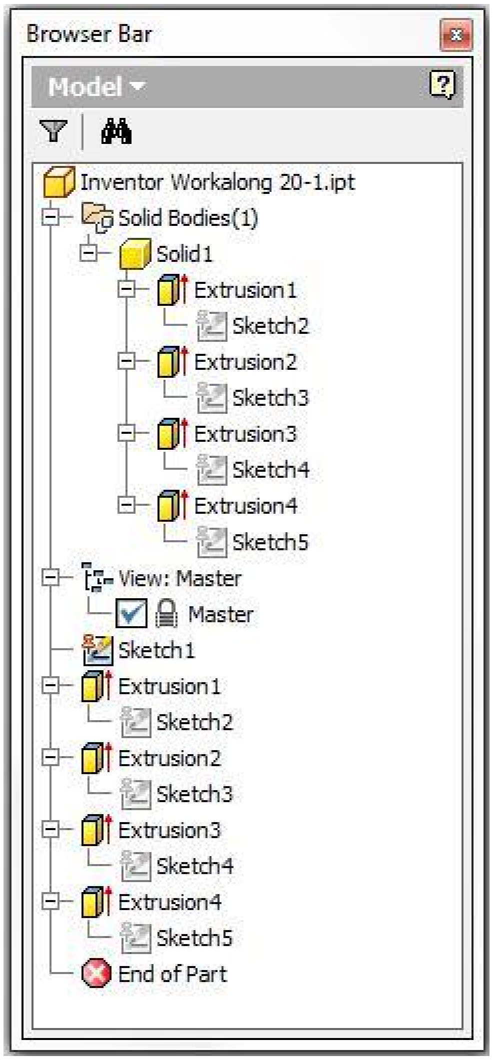


Figure Step 6B

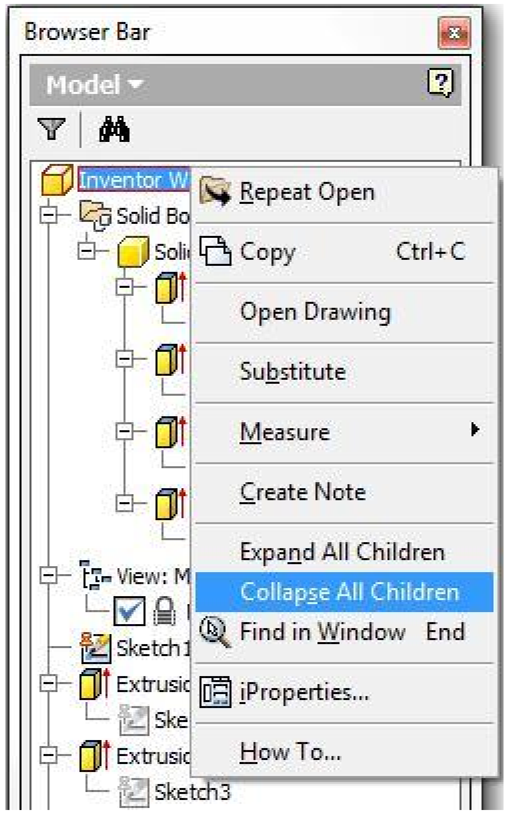


Figure Step 6C

Step 7

Right-click the part feature Extrusion1. In the Right-click menu, click Show Dimensions. Note how the 2D sketch dimensions and the extrusion dimension will display on the model. (Figure Step 7)

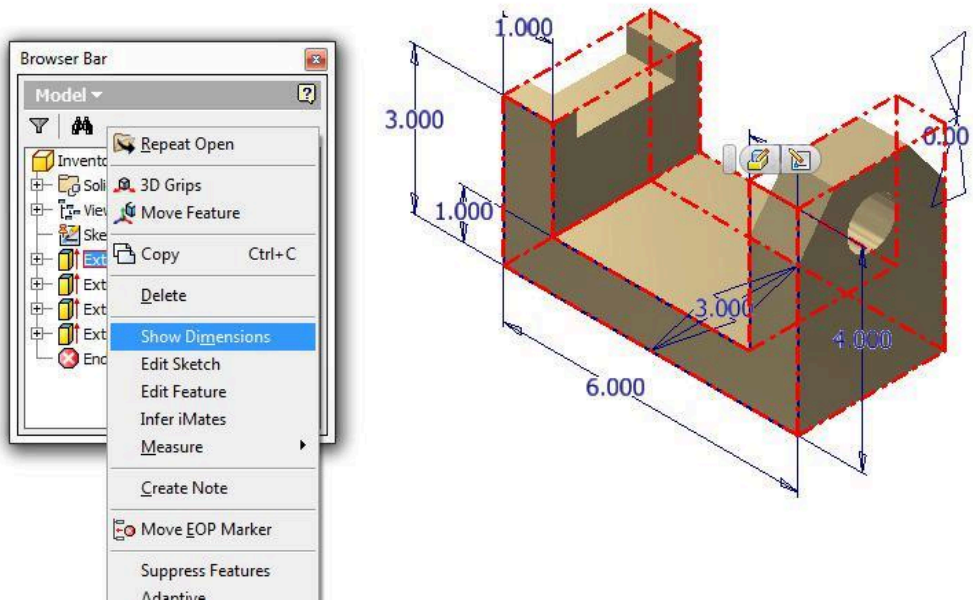


Figure Step 7 [Click to see image full size]

Step 8

Right-click Extrusion1 again. In the Right-click menu, click Edit Sketch. The Graphic window will change to Sketch mode and display the sketch. (Figure Step 8)

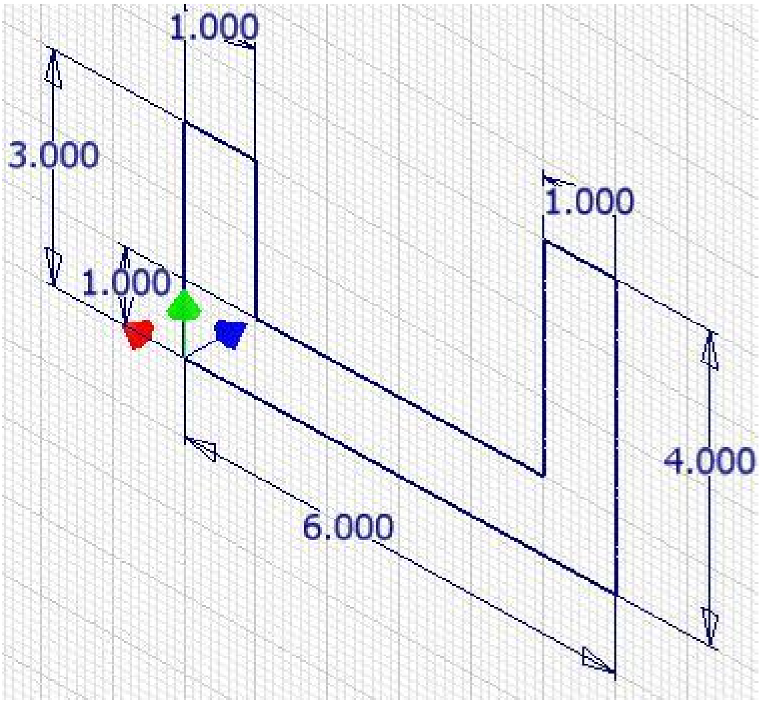


Figure Step 8

Step 9

Double click any dimension in the sketch and the Edit Dimension dialogue box will open as shown in the figure. You could now change the dimension, if required, which in turn will change the size of the model. Close the dialogue box without making any changes and return to Model mode. (Figure Step 9)

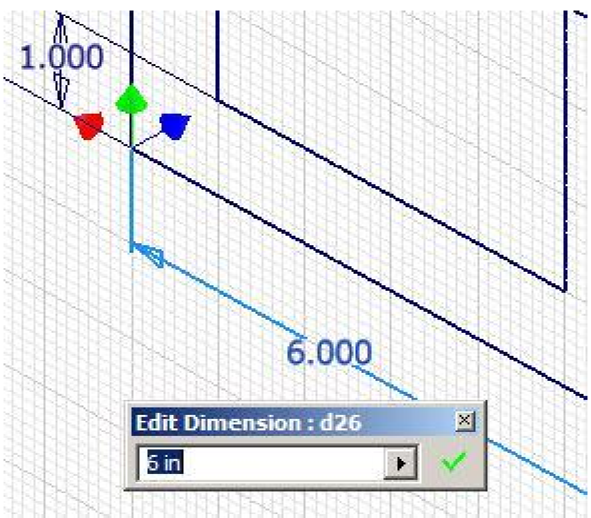


Figure Step 9

Step 10

Right-click Extrusion1. In the right-click menu, click Edit Feature. Since this feature is an extrusion, the Extrude dialogue box will open. If required, the extrude feature could be edited. Click the Cancel button to close it. (Figure Step 10A and 10B)

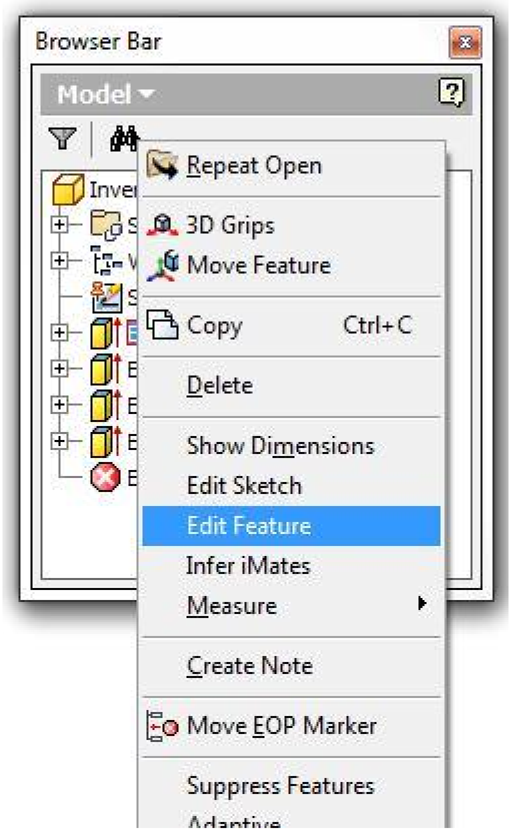


Figure Step 10A

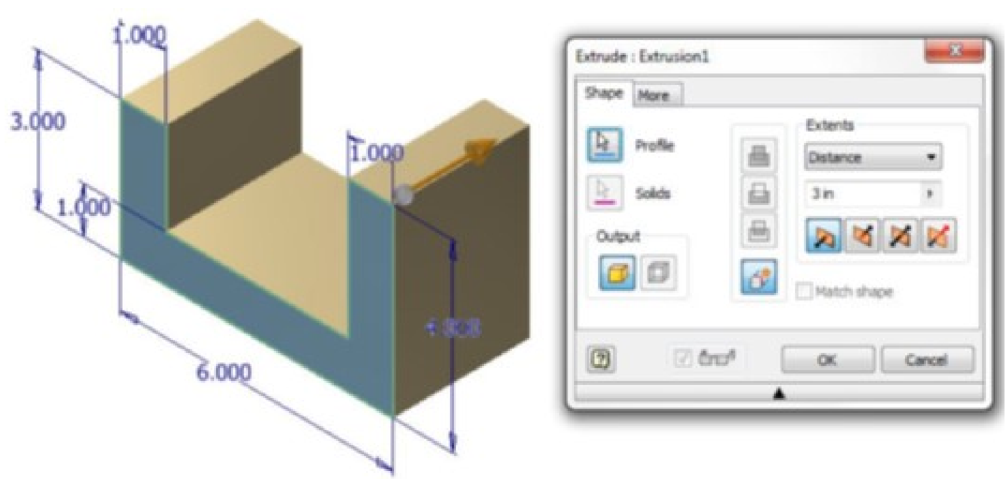


Figure Step 10B [Click to see image full size]

Step 11

Right-click Extrusion3 as shown in the figure. In the right-click menu, click Find in Window. Note how the extrude on the left side will fully display in the window. This is a handy feature to use when your part is complicated or if you are using a part you are not familiar with. (Figure Step 11)

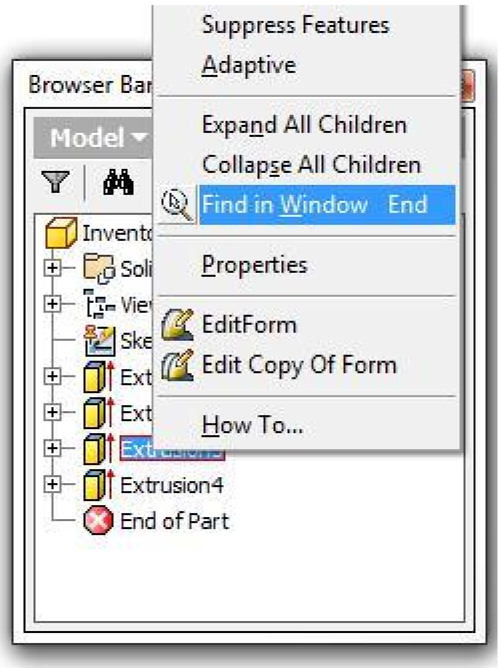


Figure Step 11

Step 12

Using what you learned in Steps 8 to 11, edit the sketch in Extrusion1 to match the dimensions shown in the figure. When you are done, click Finish Sketch to return to Model mode and the model will resize to reflect the changes you made to the dimensions. (Figure Step 12)

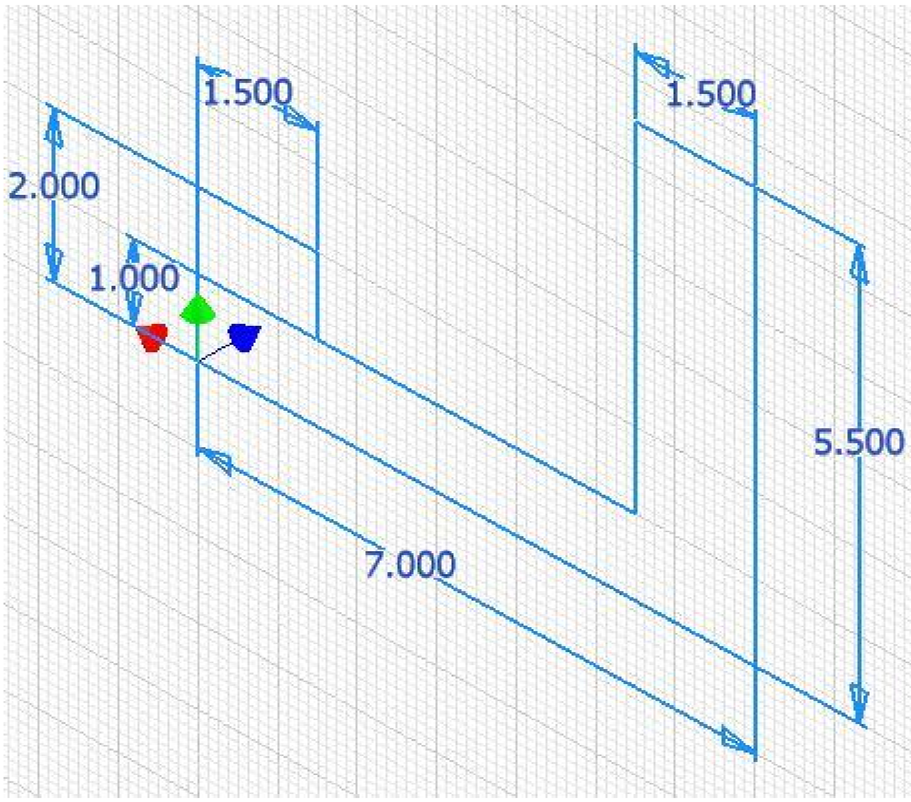


Figure Step 12

Step 13

Edit Extrusion1 to change the extrusion depth from 3 inches to 4 inches. Click OK. (Figure Step 13)

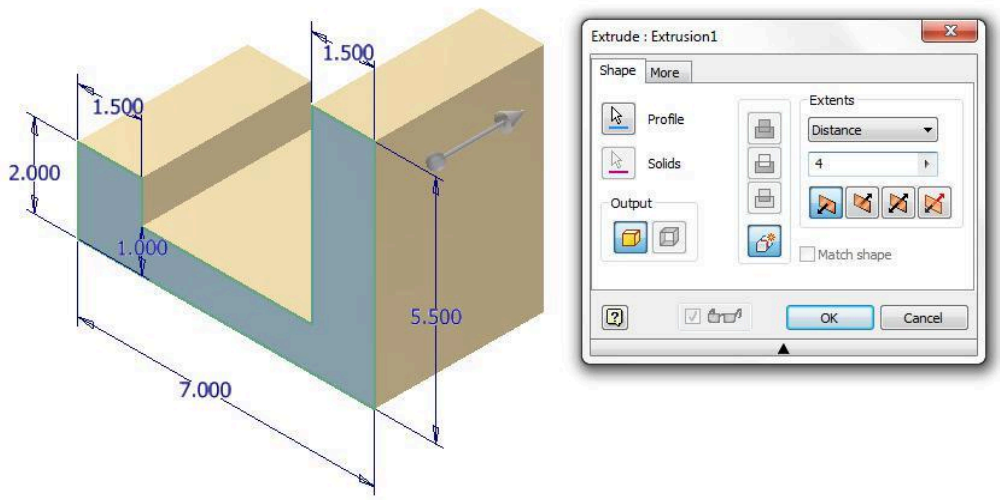


Figure Step 13 [Click to see image full size]

Step 14

Edit the dimensions in the sketch for the part feature Extrusion4 to the dimensions shown in the figure. Return to Model mode. (Figure Step 14A and 14B)

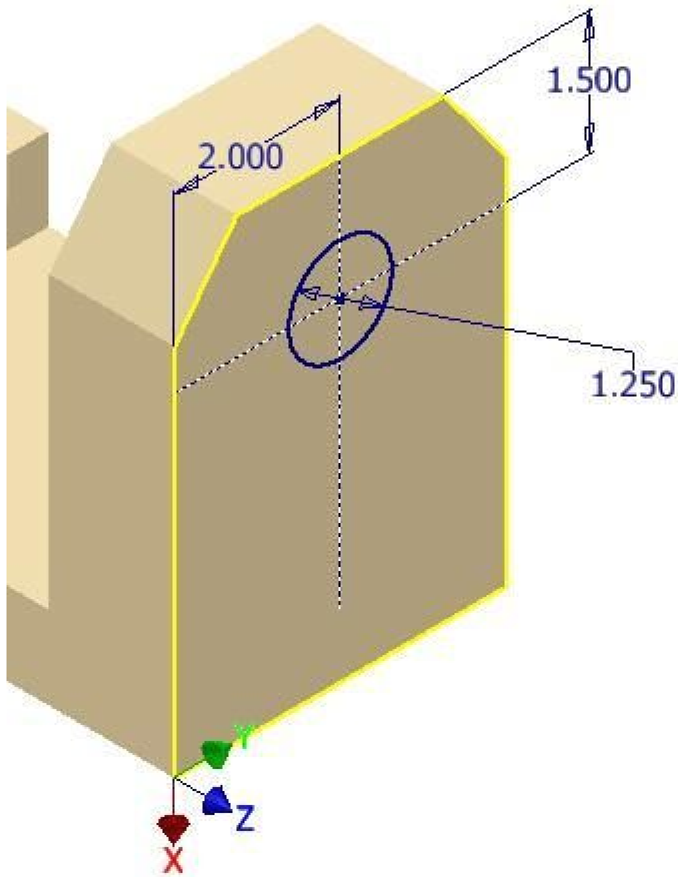


Figure Step 14A

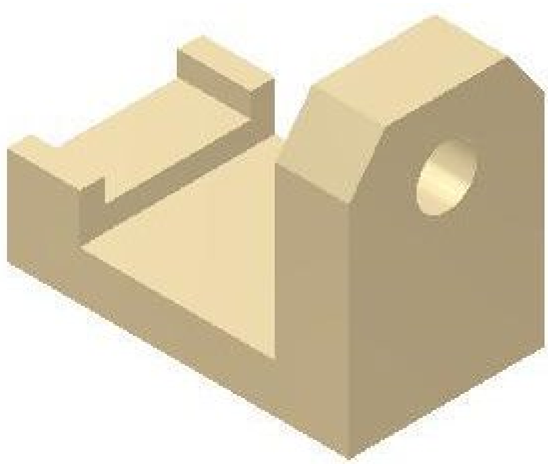


Figure Step 14B

Step 15

Since Sketch1 is an unconsumed sketch, it can be deleted. Select it and right click it. In the Right-click menu, select Delete. (Figure Step 15)

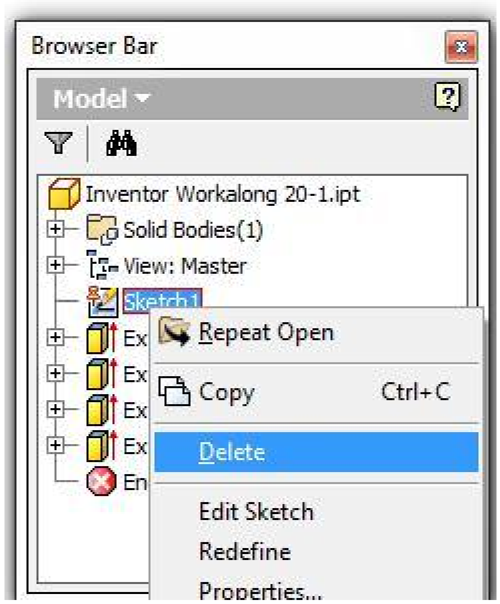


Figure Step 15

AUTHOR'S COMMENTS: The names of the part features in the Browser bar can be renamed to names that make more sense to the user. To change a name, click the existing name twice in the Browser bar and it will change to edit mode.

Step 16

Expand the children. Change the name of Sketch2 to Base Sketch – Front View. (Figure Step 16A and 16B)

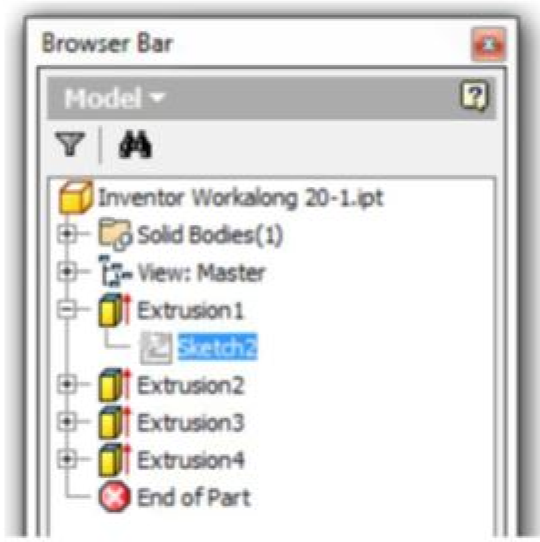


Figure Step 16A

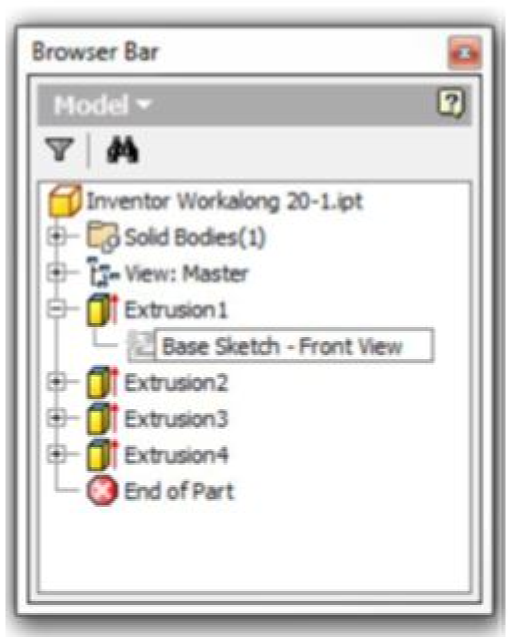


Figure Step 16B

Step 17

Change the names of all the part features and their children to match the names in the figure. Change the name of the part to: Inventor Workalong 20-1 Finished.ipt as shown in the figure. (Figure Step 17)

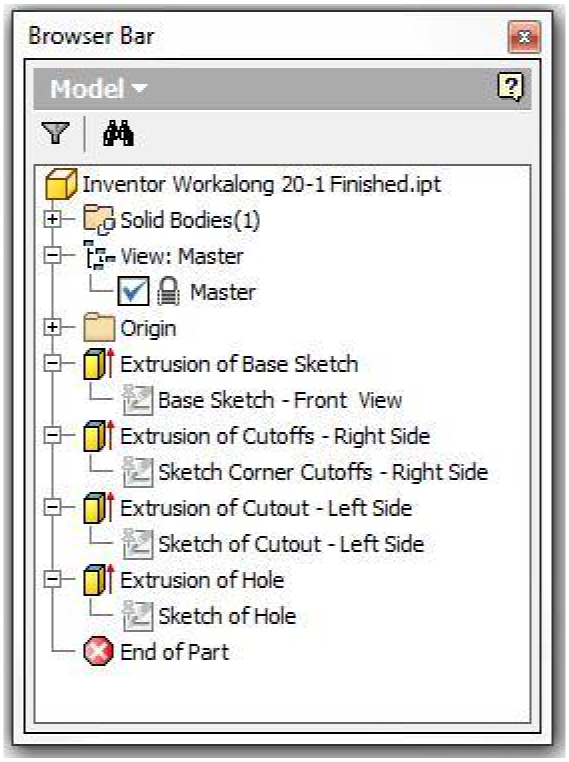


Figure Step 17

AUTHOR'S COMMENTS: When you edit the name of the file in the Browser bar, it will NOT change the name of the current file. When you save the file, it will be saved with it original name.

AUTHOR'S COMMENTS: At the end of the part features in the Browser bar is the End of Part icon. Inventor will stop displaying the model when it finds this icon in the part features list. The icon can be dragged to any location desired.

Step 18

Move the cursor onto the End of Part icon and hold down the left mouse button. When the End of Part has a red rectangle around it, drag it to the location shown in the figure. (Figure Step 18A, 18B, and 18C)

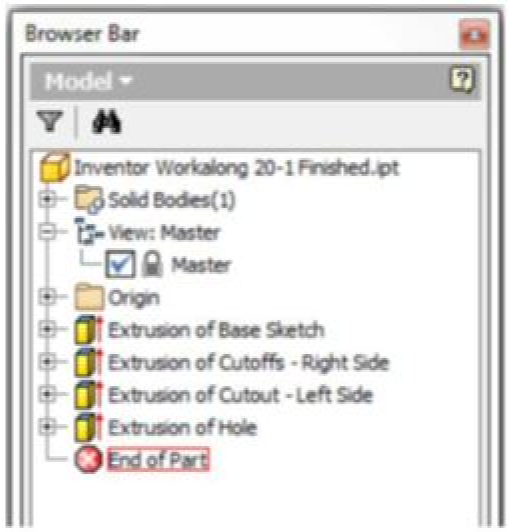


Figure Step 18A

Figure Step 18B

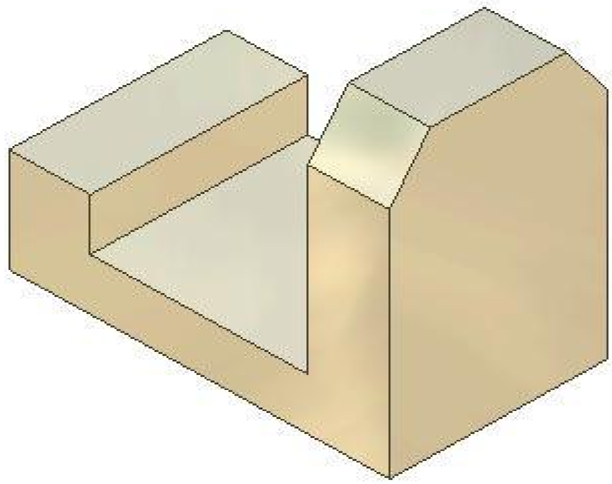


Figure Step 18C

Step 19

Drag the End of Part icon back to the bottom of the part feature list in the Browser bar. (Figure Step 19A and 19B)

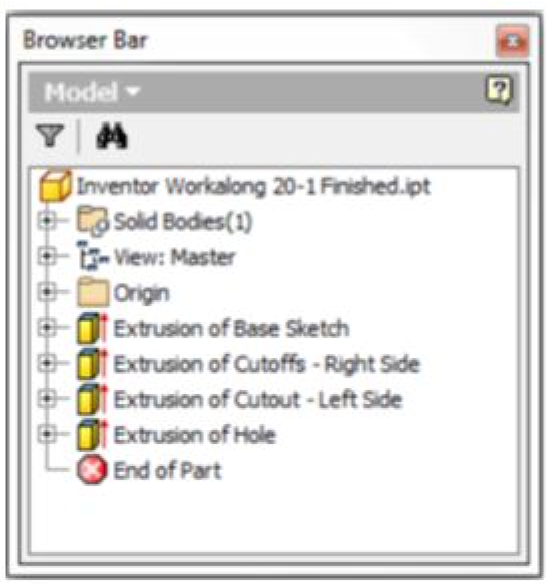


Figure Step 19A

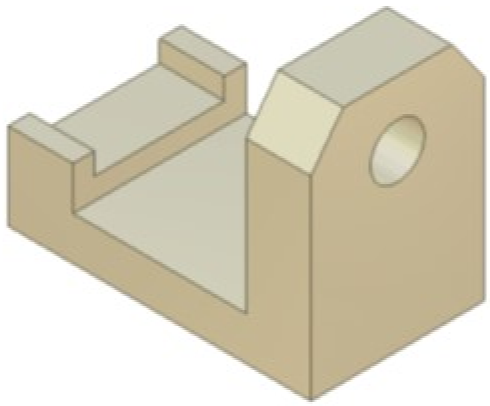


Figure Step 19B

Step 20

Using the SAVEAS command, save the file with the name: Inventor Workalong 20-1 Finished. Close the part.

Measuring the Model

Inventor has four commands available to measure distances, angles, loops lengths and areas. They also allow the user to set the units and decimal precision of how it displays the answers. Answers can be accumulated. Measurements can be made in either 3D (Model mode) or 2D (Sketch mode).

Inventor Command: MEASURE DISTANCE, MEASURE ANGLE, MEASURE LOOP, MEASURE AREA

The MEASURE command is used to measure distances, angles, loops lengths and areas. It can be used in model mode or sketch mode. If it is used in Model mode, it measures in 3D and in Sketch mode, it measures in 2D.

Shortcut: **None**

WORK ALONG: Measuring a Solid Model

Step 1

Open part file: Inventor Workalong 20-1.ipt. (Figure Step 1)

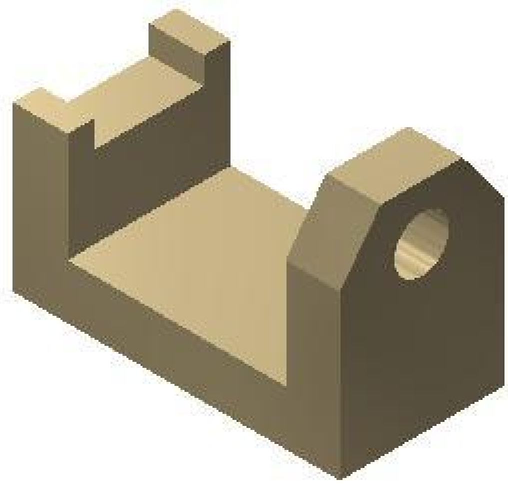


Figure Step 1

Step 2

Click the MEASURE DISTANCE command. The Measure Distance dialogue box will display as shown in the figure. Move this dialogue box close to your model to make measuring easier. (Figure Step 2).

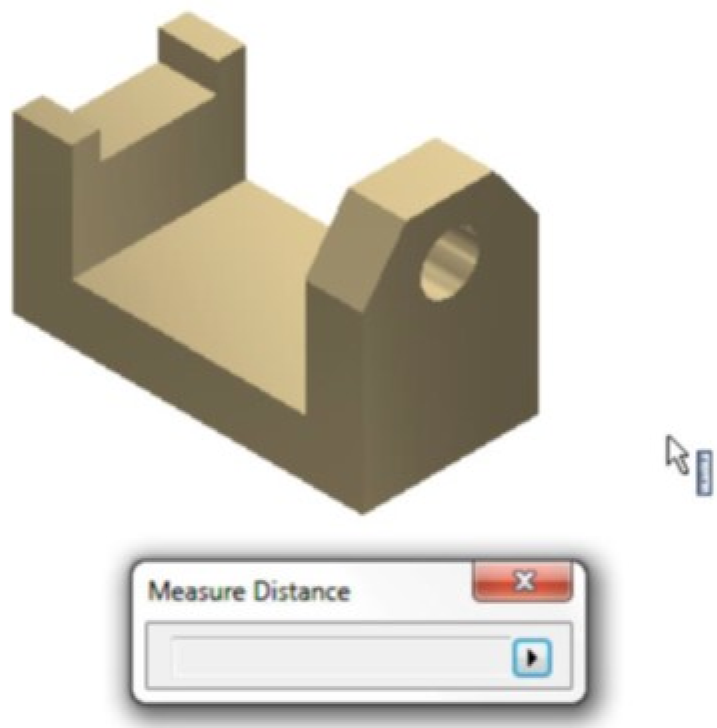


Figure Step 2

Step 3

Note that when you are in Measuring mode, the Graphic cursor appears as a ruler icon. The ruler icon indicates that you are measuring in document units. This will be covered in greater detail later in the module. (Figure Step 3)

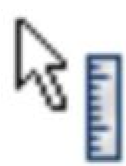


Figure Step 3

Step 4

Move the cursor to the bottom left corner of the model and click the left mouse button. Note how it displays a small snap circle indicating that you are snapping to an exact endpoint. (Figure Step 4)

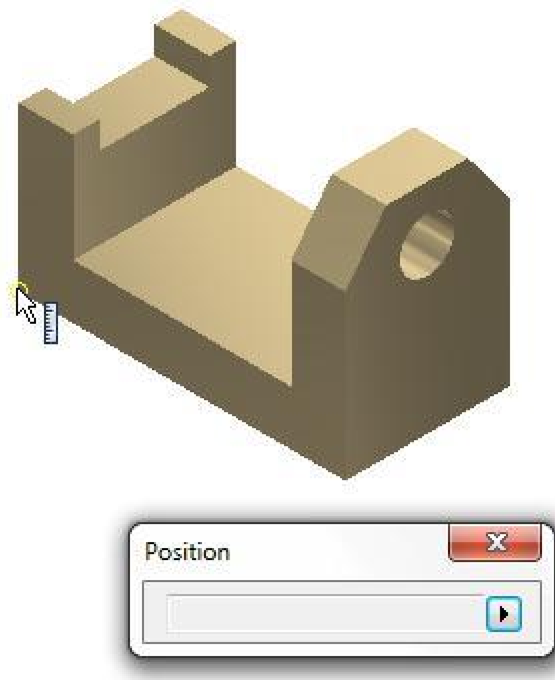
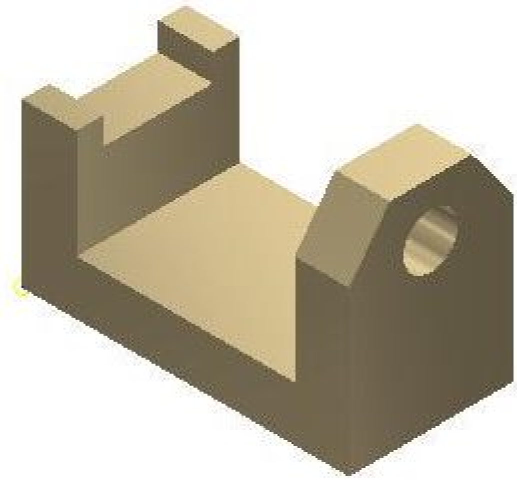


Figure Step 4

Step 5

When you snap to a location on the model, the XYZ coordinate location will display as shown in the figure. In this case, it is X0Y0Z0 of the model or the Center Point. (Figure Step 5)



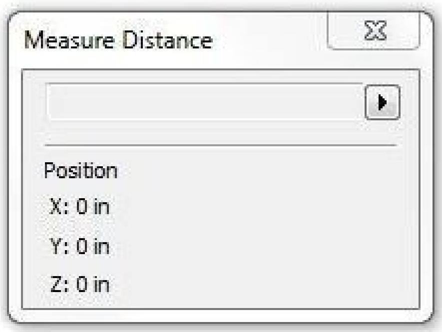


Figure Step 5

Step 6

To clear the Measure Distance dialogue box and start a new measurement, click the small triangle inside the dialogue box. In the pull-down menu, select Restart. (Figure Step 6)

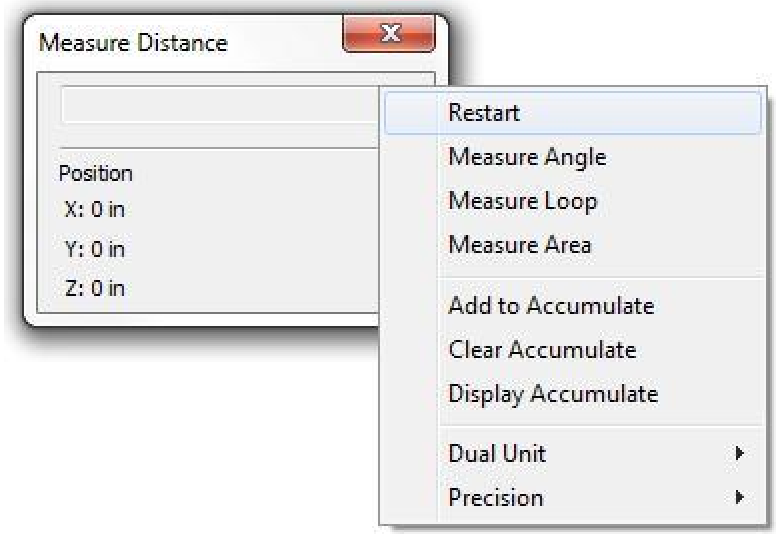


Figure Step 6

Step 7

Move the cursor to bottom right corner of the right side of the model. Click it to snap to the corner. The XYZ location will display in the dialogue box as shown in the figure. (Figure Step 7)

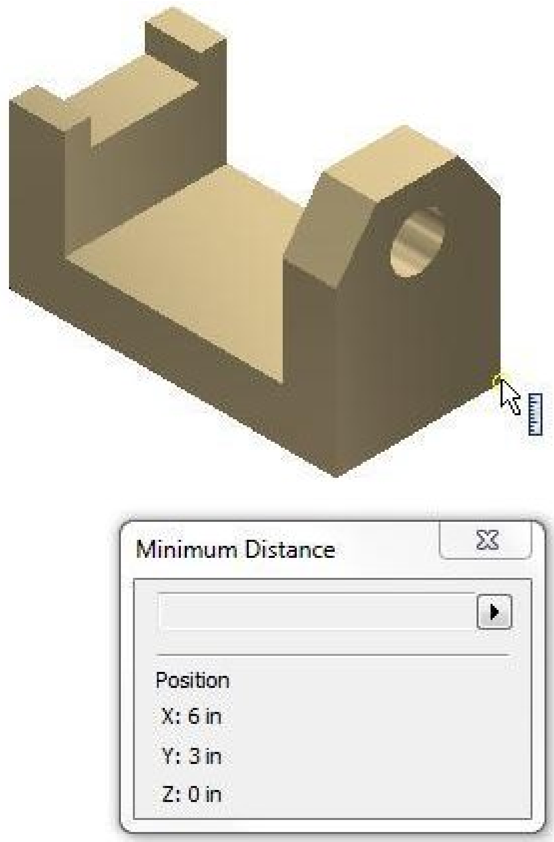


Figure Step 7

Step 8

Clear the dialogue box by clicking Restart. Measure the length of an edge. Move the cursor onto the left edge of the right side of the model as shown in the figure. (Figure Step 8)

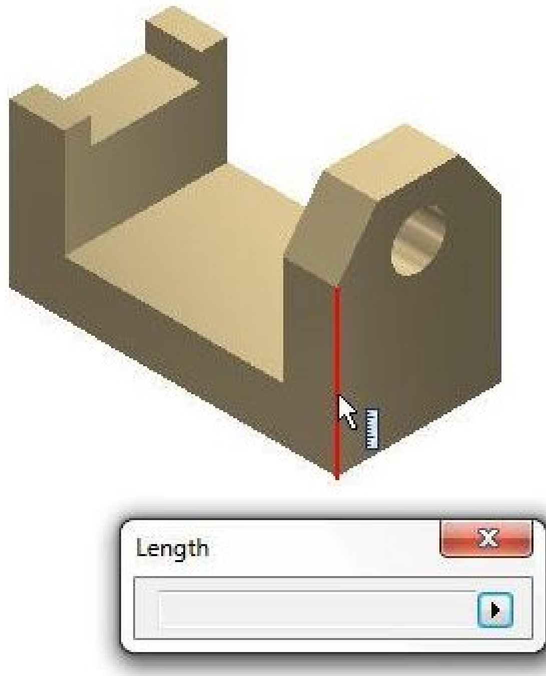


Figure Step 8

Step 9

When you click the mouse, the length of the edge will display in the dialogue box. In this case, it is 3 inches long. (Figure Step 9)

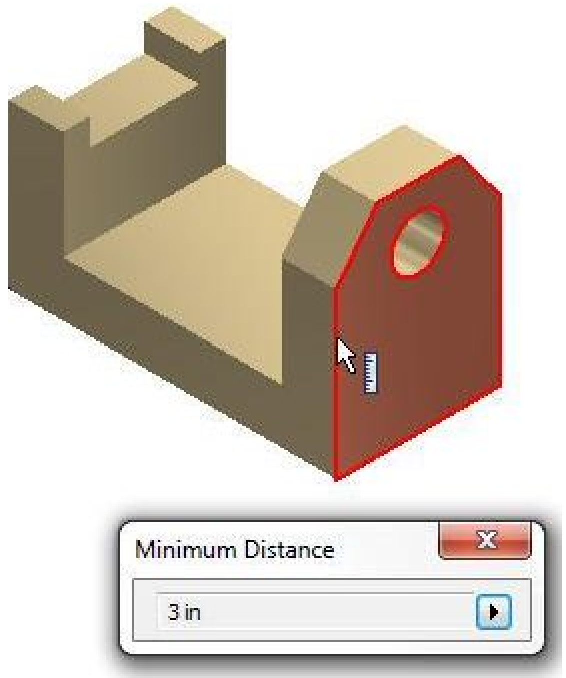


Figure Step 9

Step 10

While the dialogue box is still displaying the length of the edge, click the opposite edge. Note how the dialogue box now displays both the length of the edge and the distance between the edges. A line showing the measured distance will display to show you what is being measured. (Figure Step 10)

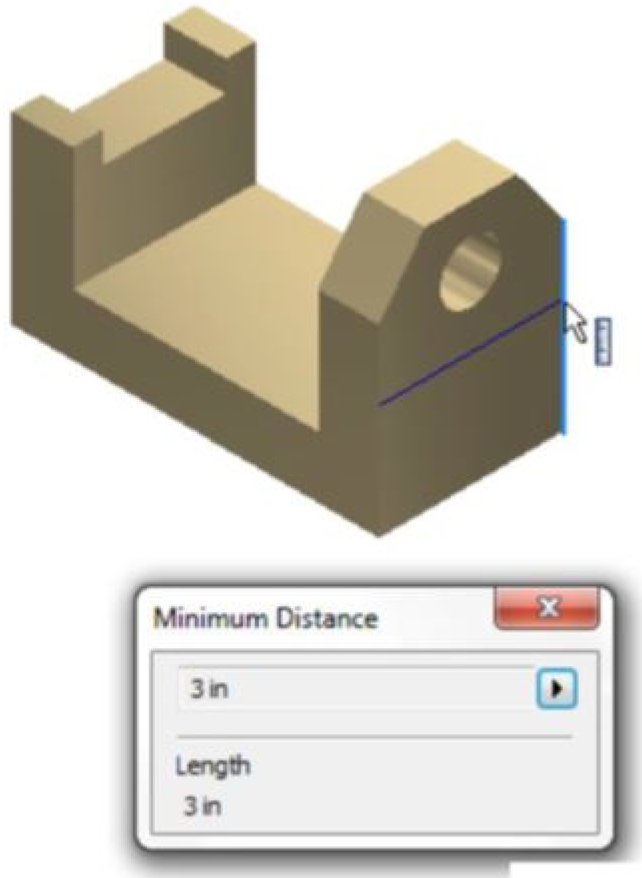


Figure Step 10

Step 11

To measure a three dimensional distance, click the two corners shown in the figure. The 3D dimension between the corners will display as well as the XYZ location of the last corner and the delta XYZ distances. (Figure Step 11A and 11B)

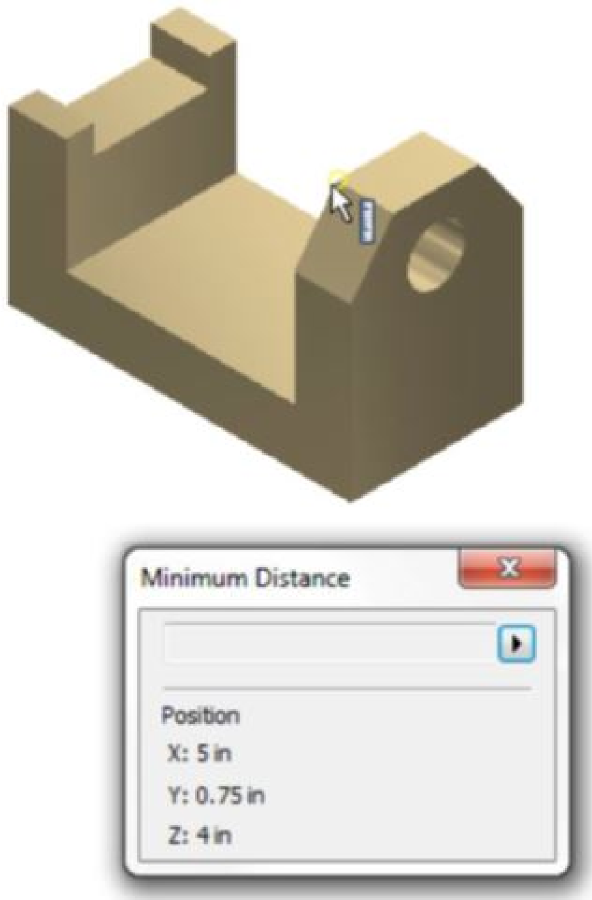


Figure Step 11A

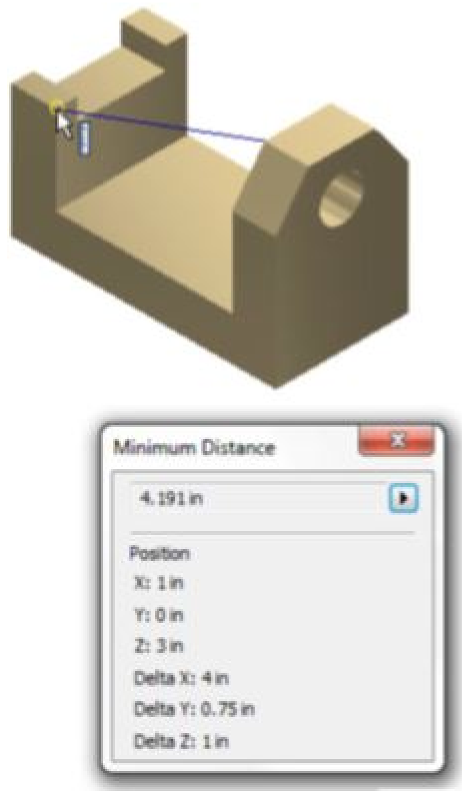


Figure Step 11B

Step 12

Measure the angle between the two edges by clicking the small triangle at the end of the dialogue box. In the pull-down menu, select Measure Angle. Select the two edges. The angle will display graphically and as an exact number in decimal degrees. (Figure Step 12A and 12B)

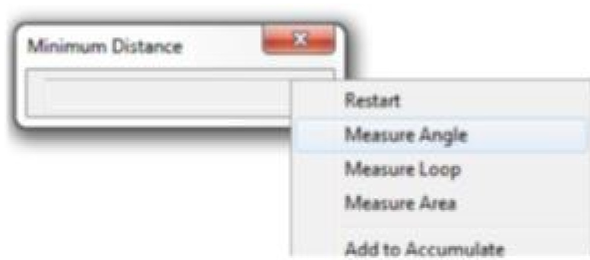


Figure Step 12A

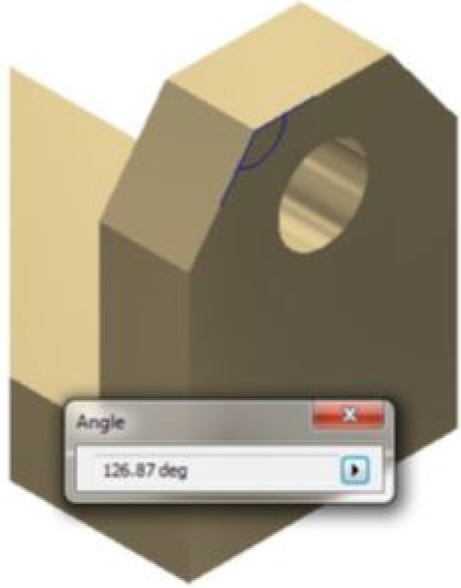


Figure Step 12B

Step 13

Change the command to Measure Loop. Click the face on the Front view. Note how the complete loop (the perimeter) around the front plane will highlight and the loop distance of 22 inches will display in the dialogue box. (Figure Step 13A and 13B)

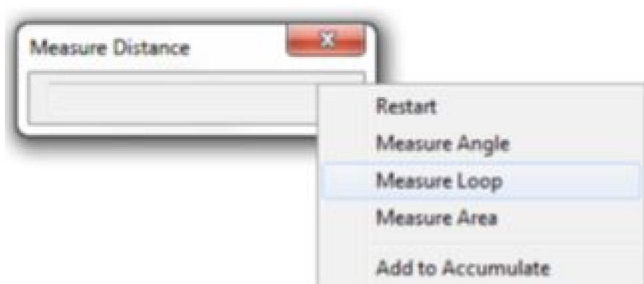


Figure Step 13A

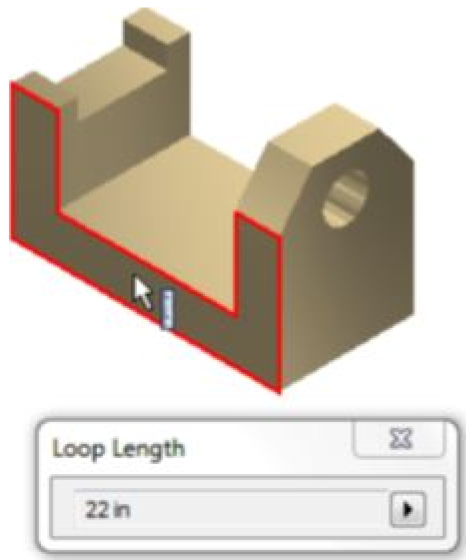


Figure Step 13B

Step 14

Measure the area of the right side face by changing the command to Measure Area. Click the face on the right side and the area to be measured will highlight. The area will display in the dialogue box. Since there is a hole in this plane, the area will be measured minus the hole. (Figure Step 14A and 14B)

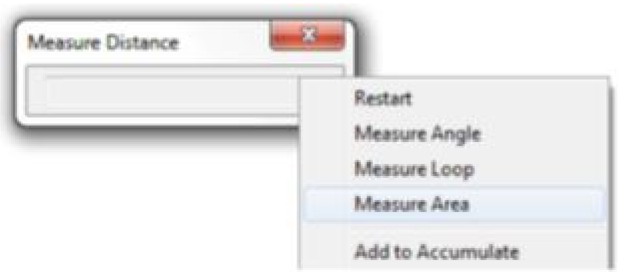


Figure Step 14A

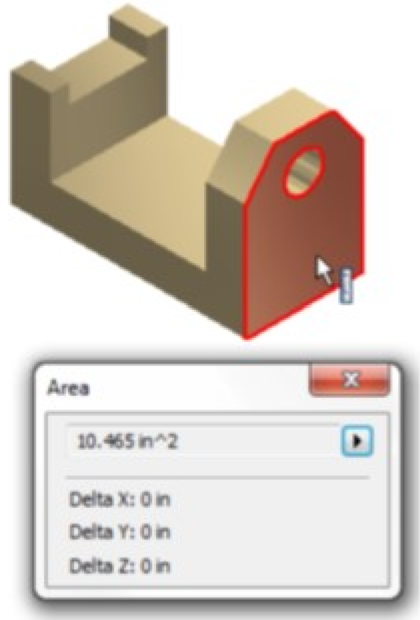


Figure Step 14B

Step 15

Suppress the hole feature on the right side face so that you can measure the area of the face without subtracting the hole. Right-click the feature in the Browser bar. In the Right-click menu, select Suppress Feature. Note how Extrusion4 is now grayed out. (Figure Step 15)

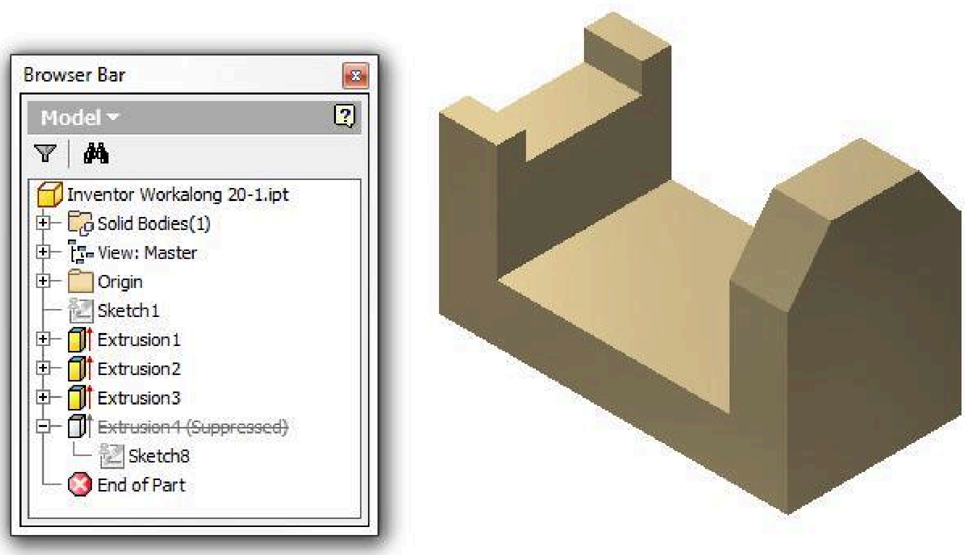


Figure Step 15

Step 16

Using what you just learned, measure the area of the right-side minus the hole. After you measure it, unsuppress the hole feature. (Figure Step 16)

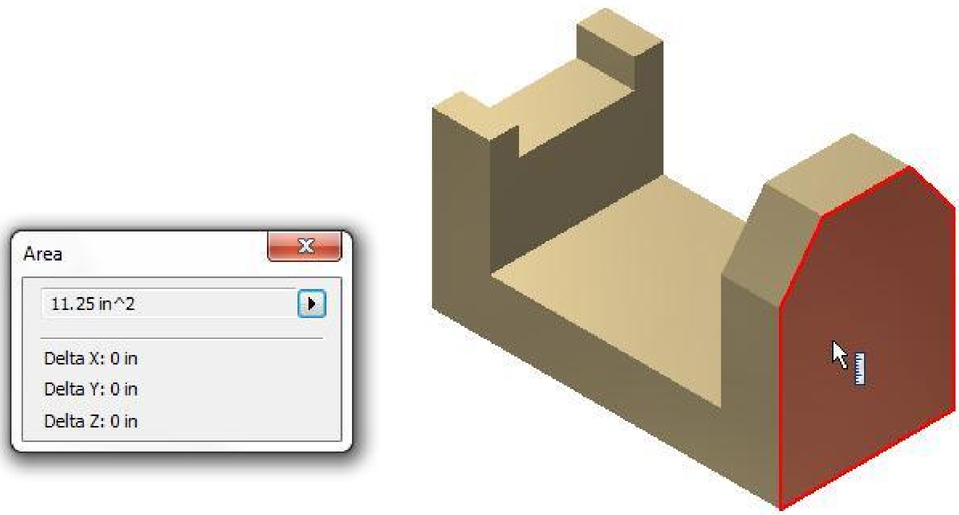


Figure Step 16

Step 17

Click Tools and then click Document Settings. This will open the Document Setting dialogue box for the current part. Do not make any changes at this time. Click Close to close the dialogue box. (Figure Step 17A and 17B)

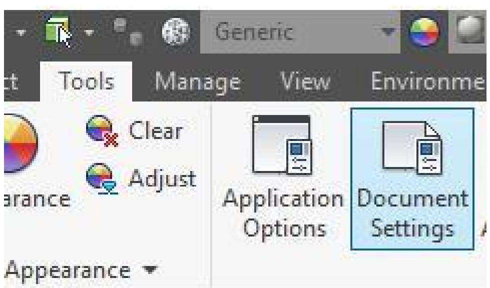


Figure Step 17A

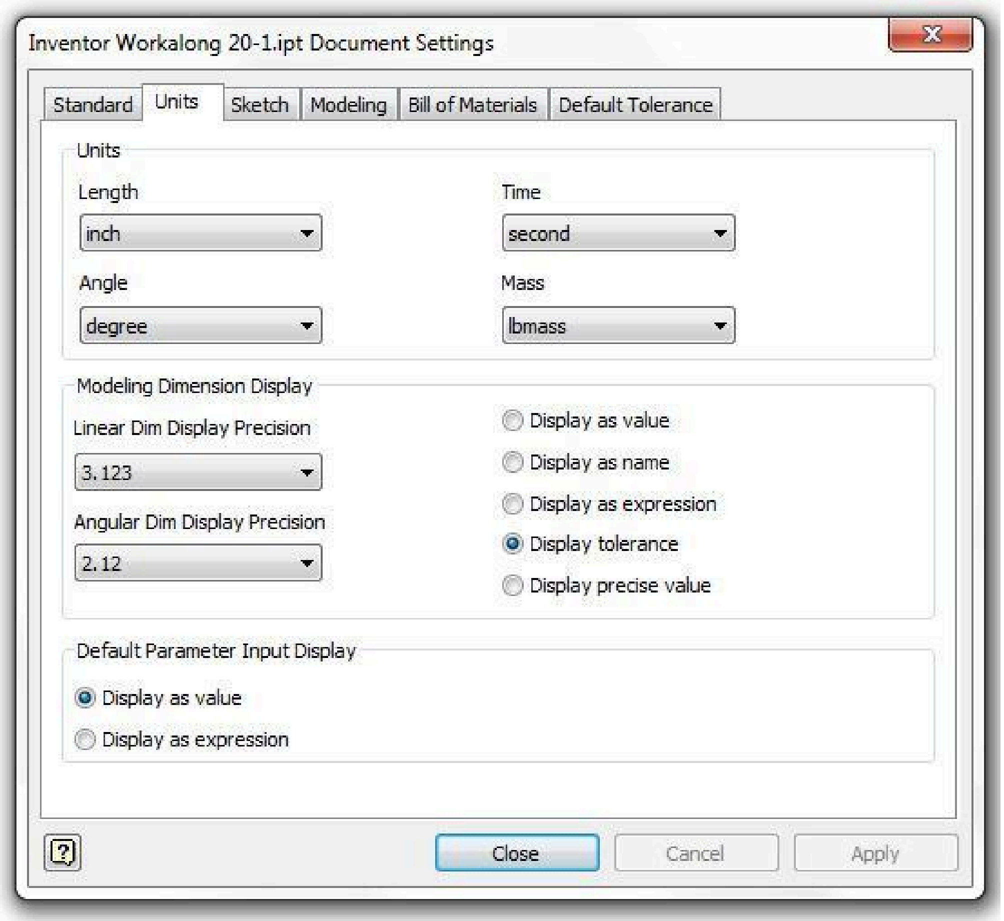


Figure Step 17B

AUTHOR'S COMMENTS: The units and precision that Inventor has been displaying to your measurement commands have been preset in the document settings. Document settings are preset in the template file that you selected when you started the part. If required, you can make changes to the settings.

Step 18

Enter the MEASURE command. Click the small triangle at the end. In the pull-down menu, select Precision and then 5 places. An example of an area measurement with a precision of 5 decimals points is shown in the figure.

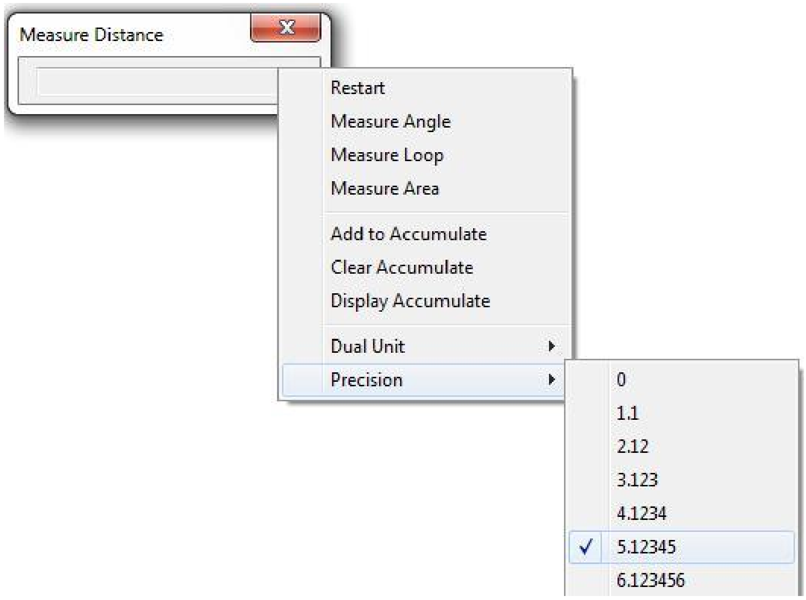


Figure Step 18

Step 19

Change the precision to display All Decimals. A sample of an answer to an area measurement is shown in the figure. (Figure Step 19)

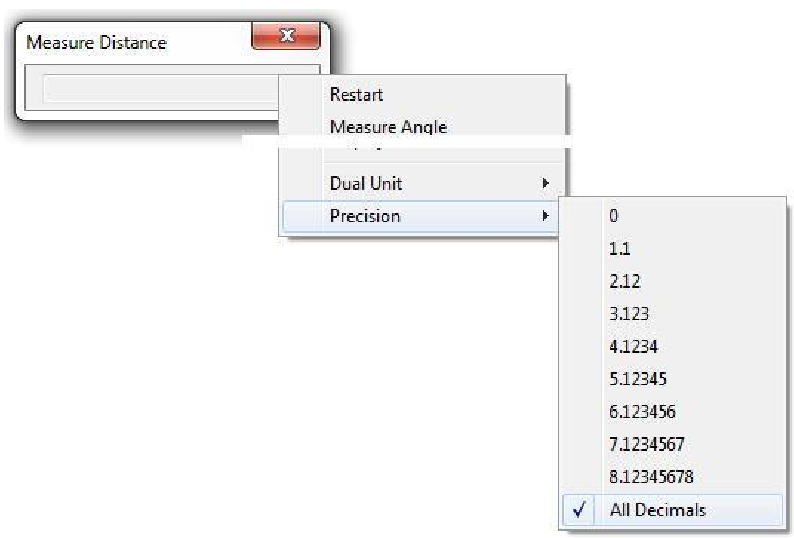


Figure Step 19

AUTHOR'S COMMENTS: When the precision is set to All Decimals, the Graphic cursor will change to the Micrometer icon as shown in Figure below.

Step 20

Change the precision back to three decimal places. (Figure Step 20)

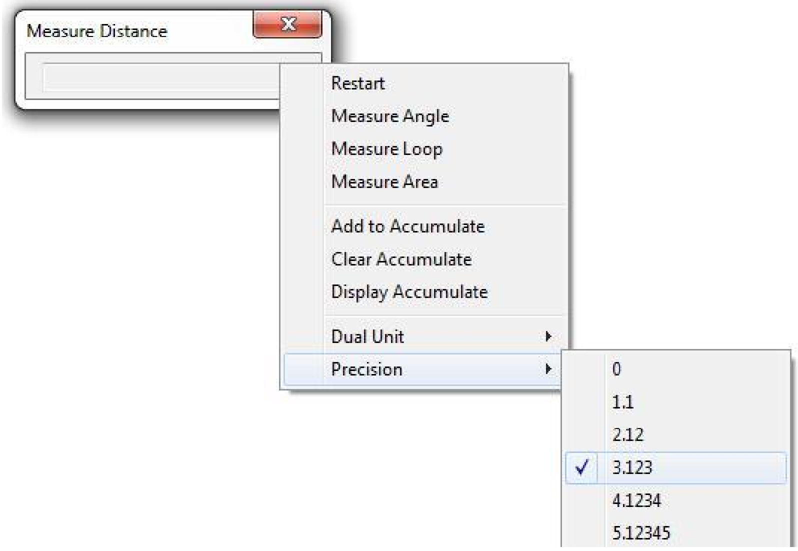


Figure Step 20

Step 21

Change the colour to: Aluminum Cast. (Figure Step 21)

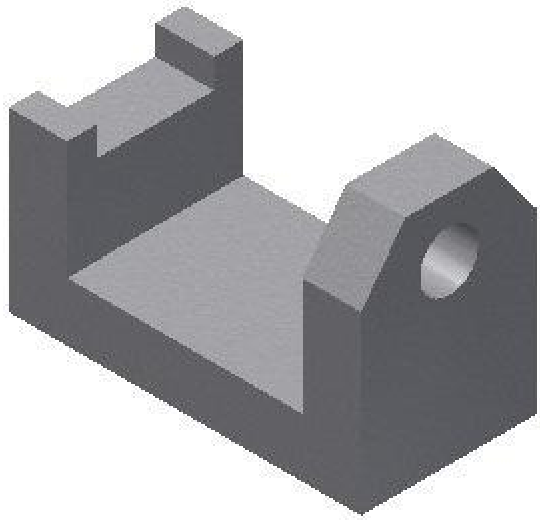


Figure Step 21

Step 22

Save and close the part.

Geometry Lesson: Physical Properties

Mass:	the quantity of the matter contained in the solid model. This is determined by multiplying the volume of the solid times the density of the material it is made from. Mass is not dependent on gravity which makes it different but proportional to the weight. Mass is used when considering a measure of a solid's resistance to inertia.
Volume:	The amount of space occupied by the solid model.
Density:	The weight of material usually expressed per cubic inch or cubic millimeter. i.e. 21.55^3in or 42.87^3mm
Center of Gravity:	Geometrical centre of the solid model. It is also called the centroid. If the density of the solid is uniform, the centre of gravity or centroid is located at the centre of the mass.
Mass =	Volume X Density Expressed as lbmass, grams or kg
Area =	Length X Width of a 2D plane. Expressed as sq in or sq mm. ie. 10.5^2in or 23.5^2mm
Volume –	cubic inches or millimeters. Expressed as cu in or cu mm ie. 120.5^3in or 55.3^3mm
lbmass –	pounds
kg –	kilograms 1 kg = 1000 grams 1 kg = 2.2 pounds
in –	inches
mm –	millimeters 1 in = 25.4 mm
Inventor will display answers in scientific notation to control the number of decimals displayed:	For Example 1,250,000 = 1.25E+006 (move the decimal 6 places positively) 0.0001 = 1.0E-004 (move the decimal 4 places negatively)

Face Coloring

Sometimes it is helpful to be able to change the colour of a face. This is especially helpful when working with complex parts or to display different textures on a model. For example, this can happen during manufacturing when a part is cast and has machined surfaces. By applying the cast material to the part and then changing the faces of the machined areas to a polished texture/color, the model will appear more realistic.

A face colour overrides the part colour and, if applicable, the feature colour for selected faces. In this module, only faces coloring will be taught. The general rules when changing the colour of faces are:

1. A face colour overrides the part feature colour which in turn overrides the part colour.
2. If the part has been assigned a transparent colour, the face colour you apply will change the colour but it will be transparent.

3. If a thread texture is applied to the feature, a change to the face colour affects the base colour used in the thread texture.
4. If the face colour has been changed and the feature is colored as a pattern, the patterned features will not display the face colour.

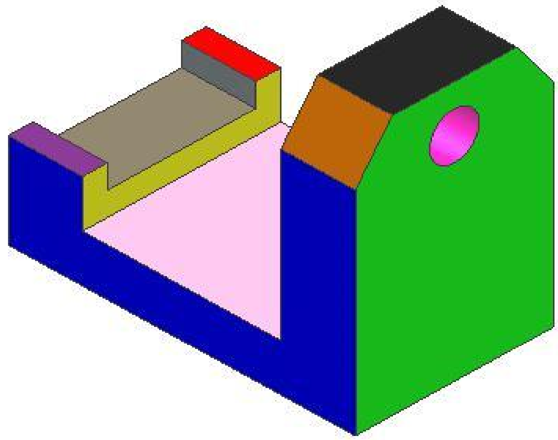


Figure 20-3
Edges and Faces

Physical Properties

The *physical properties* of a solid model include the mass, volume, centre of gravity, and inertial properties. Using the physical properties helps you to evaluate how the designed model correlates to its physical counterpart. For example, the weight of a solid model made from different materials could easily be found.

Assigning the Material to the Model

Up to this point in the book, only colour or texture has been assigned to the solid model. In this module, assigning the actual material that the part is made from will be taught.

WORK ALONG: Coloring Faces and Working with Physical Properties

Step 1

Open part file: Inventor Workalong 20-1.ipt. (Figure Step 1)

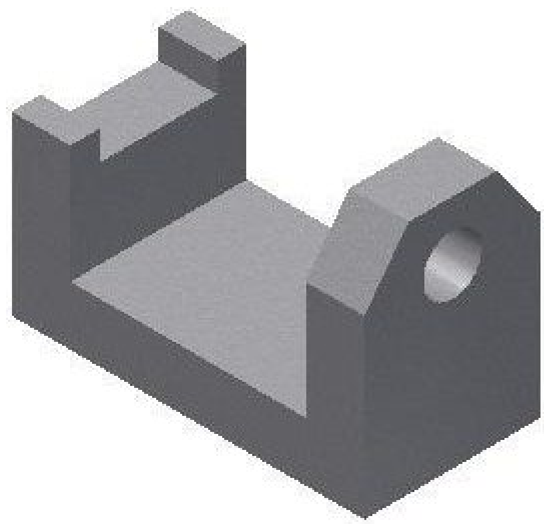


Figure Step 1

Step 2

Click the right side face to select it and right click the mouse. In the Right-click menu, select Properties. In the Face Properties dialogue box, select: Blue – Wall Paint – Glossy. (Figure Step 2A, 2B, and 2C)

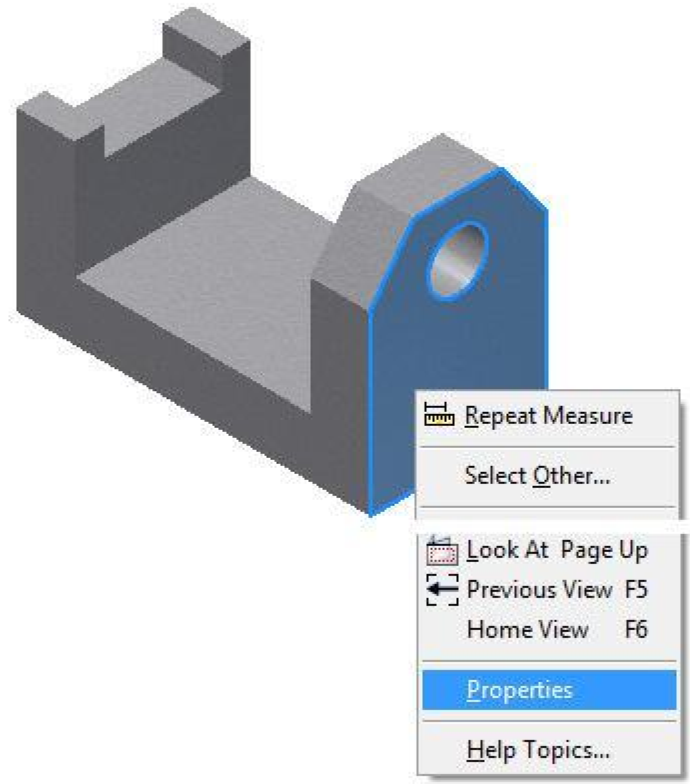


Figure Step 2A

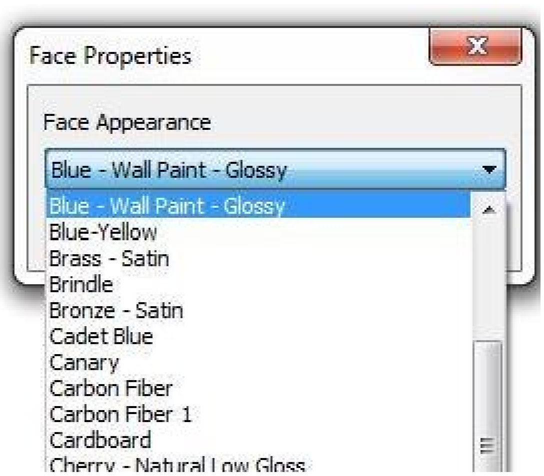


Figure Step 2B

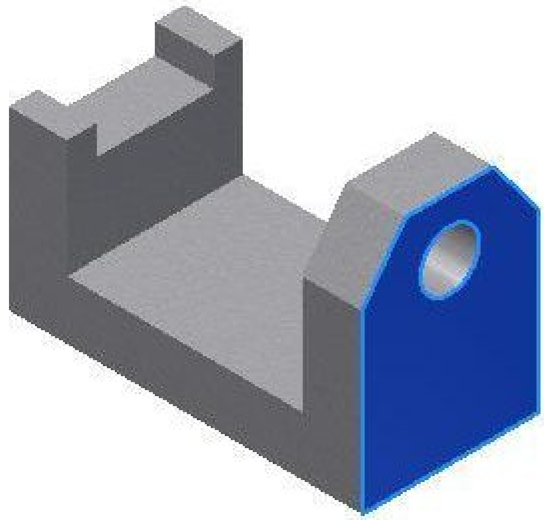


Figure Step 2C

Step 3

Using what you just learned, change three additional faces as shown in the figure to Green, Orange and Red. (Figure Step 3)

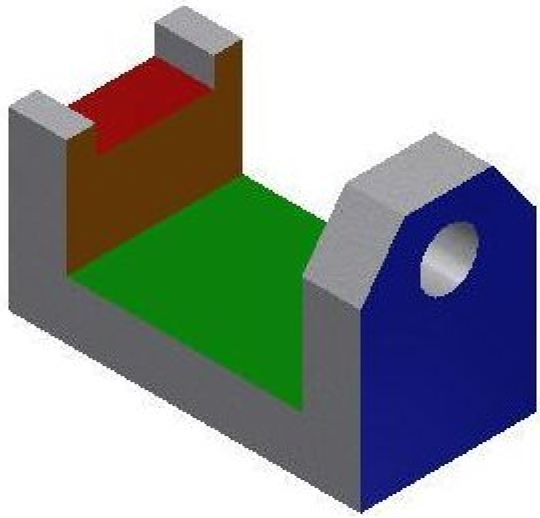


Figure Step 3

Step 4

Click the View tab. Click the CENTER OF GRAVITY command. Note how the Center of Gravity icon will display located at the centre of gravity or centroid of the model. It also displays the positive X, Y and Z axis. (Figure Step 4A and 4B)

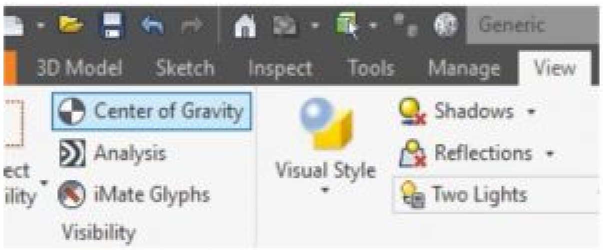


Figure Step 4A

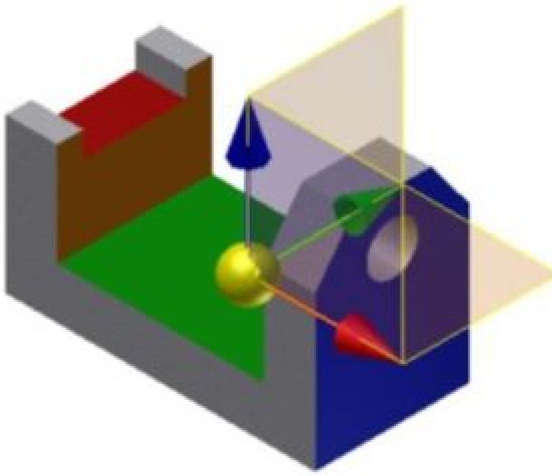


Figure Step 4B

Step 5

Rotate the model so that you can see the location of Center of Gravity icon from a better viewpoint . (Figure Step 5)

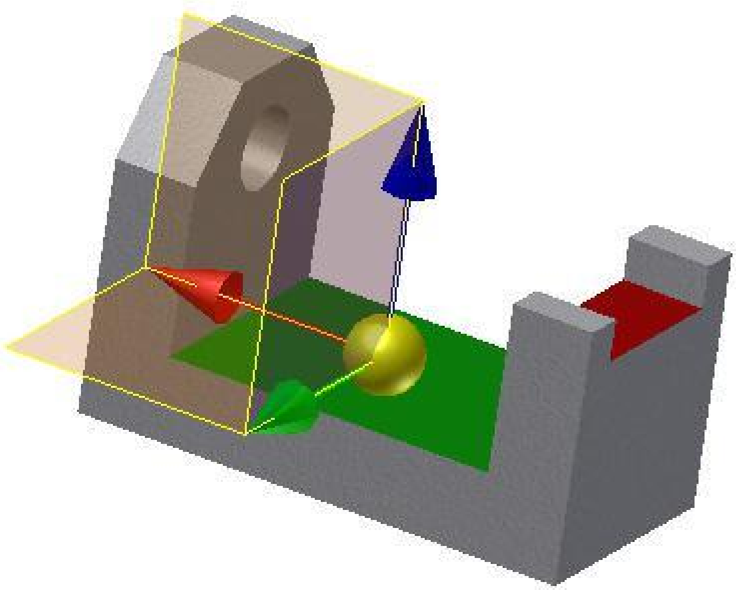


Figure Step 5

Step 6

Disable the display of the Center of Gravity icon.

Step 7

Right-click the part name in the Browser bar. In the Right-click menu, select iProperties. This will open the Properties dialogue box for the current part. (Figure step 7A and 7B)

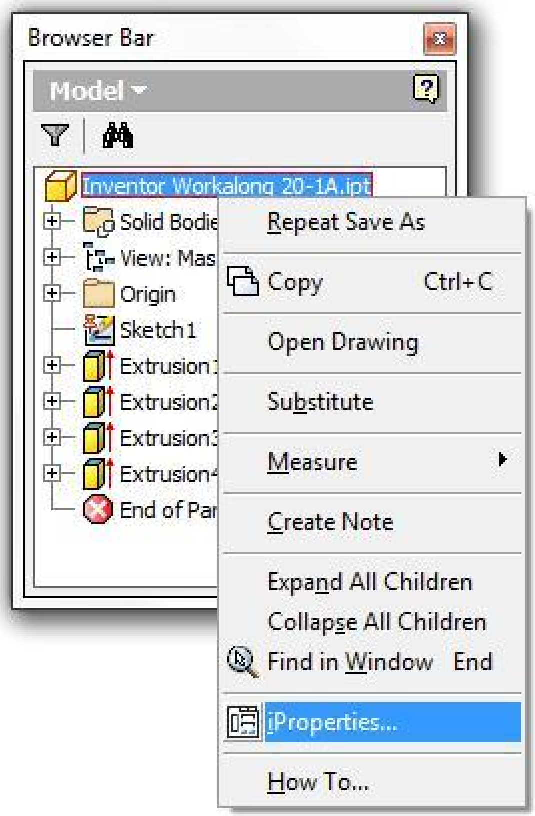


Figure Step 7A

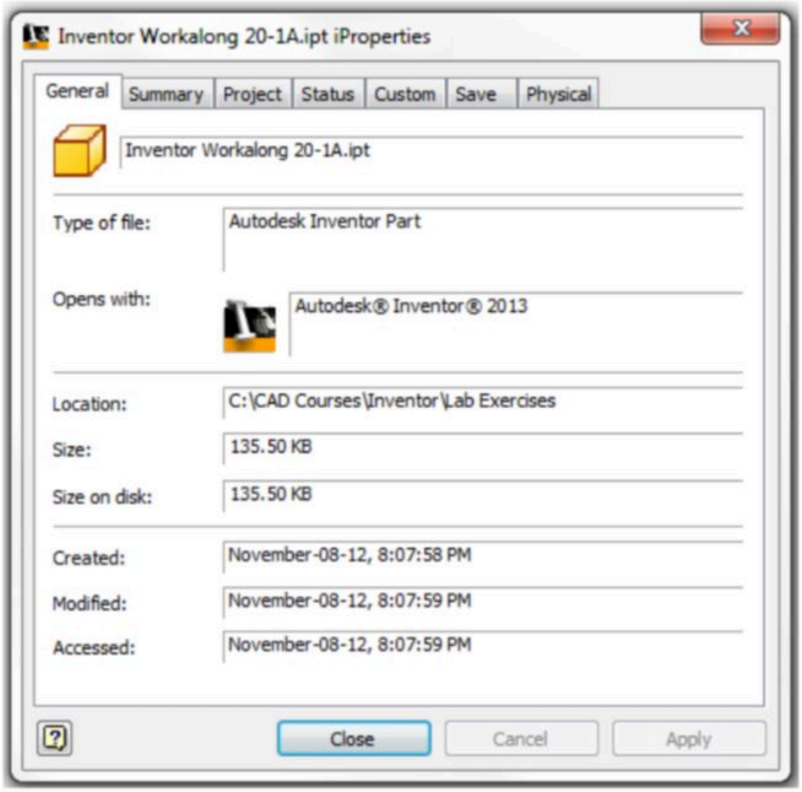


Figure Step 7B [Click to see image full size]

AUTHOR'S COMMENTS: When a Default material is assigned it means that no actual material has been assigned to the part. Even though the colour of the solid is set to Aluminum (Cast), that is only the appearance of the model. The material that the solid model is made from must also be assigned.

Step 8

Enable the Physical tab. Note how the material is shown as Default. Pull down the Material list and select: Aluminum 6061. (Figure Step 8)

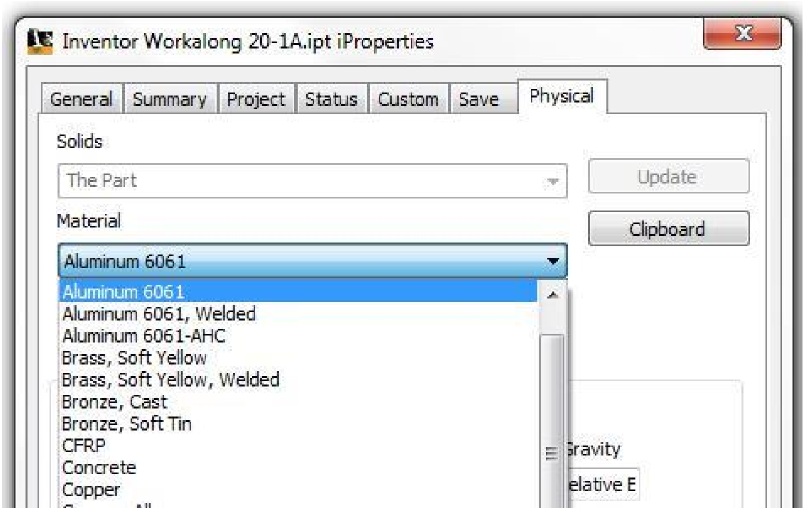


Figure Step 8

Step 9

Change the Requested Accuracy to Very High. The Density box displays the density of the material: Aluminum 6061. (Figure Step 9)

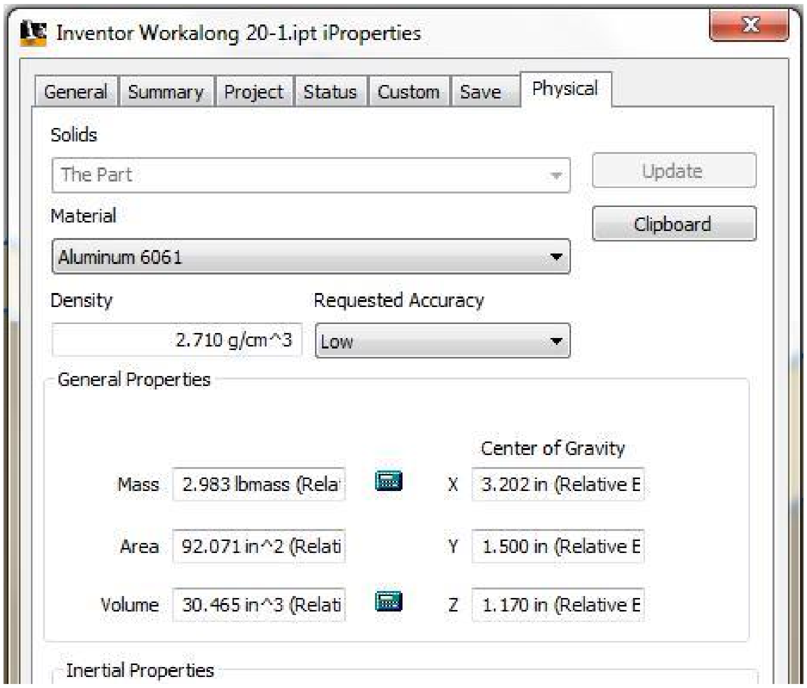


Figure Step 9

Step 10

Using what you learned earlier in the workalong, open the Document Settings dialogue box and in the Units tab, change the Mass to gram. (Figure Step 10)

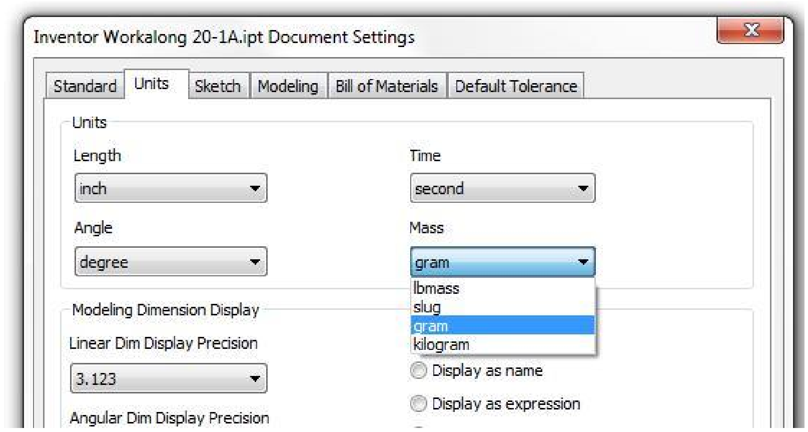


Figure Step 10

Step 11

Note how the mass (weight) is now displayed in grams. (Figure Step 11)

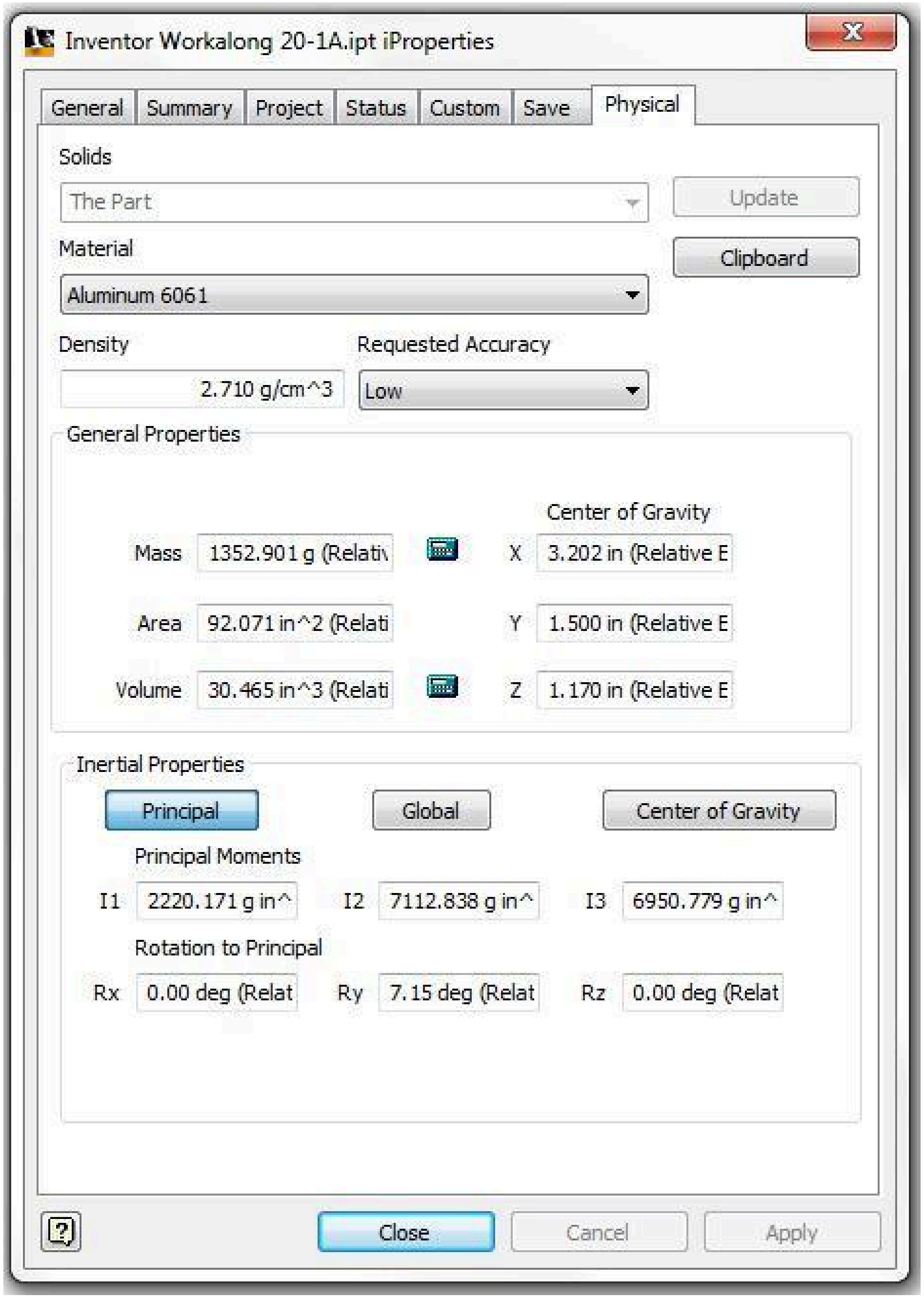


Figure Step 11

Step 12

Change the material to: <u>Cast Steel</u>. Note how the density and the mass change since steel is much denser then aluminum.

Step 13

Change the material back to: Aluminum-6061. Close the Properties dialogue box.

Step 14

Save and close the part.

Key Principles

Key Principles in Module 20

1. The Browser bar is used extensively as a tool to assist you when modifying solid models.
2. Work features are the basic sketching planes (XY, XZ, YZ), the axes (X, Y, Z) and the Center Point. Part features are the 3D features added to the solid model in model construction.
3. Edges are the lines, circles or arcs located between the planes that form the solid model. Faces are the planes between the edges of the solid.
4. Inventor has four commands available to measure distances, angles, loops lengths, and areas. They also allow you to set the units and decimal precision of how it displays the answers. Answers can be accumulated. Measurements can be made in either 3D (Model mode) or 2D (Sketch mode) mode.

Lab Exercise 20-1

Time allowed: 60 minutes.

Part Name: Inventor Lab 20-1	**Project**: Inventor Course	**Units**: Millimeters
Template: Metric – Modules Part (mm).ipt	**Color**: Aluminum (Cast)	**Material**: Aluminum – 6061

Step 1

Open the part: Inventor Lab 19-3.ipt that you created in Module 19. (Figure Step 1)

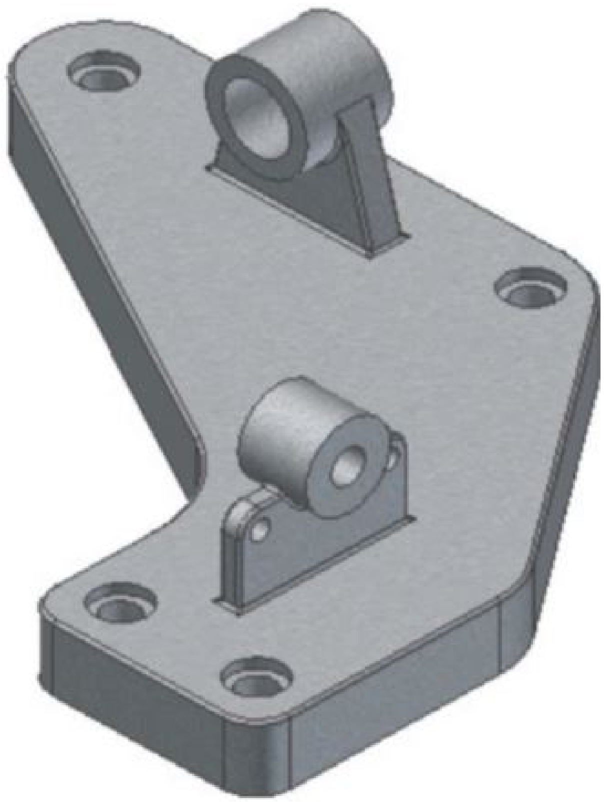


Figure Step 1
Modified Solid Model

Step 2

Save the file as: Inventor Lab 20-1.

Step 3

Change the model colour to: Aluminum (Cast).

Step 4

Expand All the Children in the Browser bar.

Step 5

Edit the Base sketch and change the dimensions of the overall length from 180 to 200 and the overall width from 140 to 175 in the figure. (Figure Step 5)

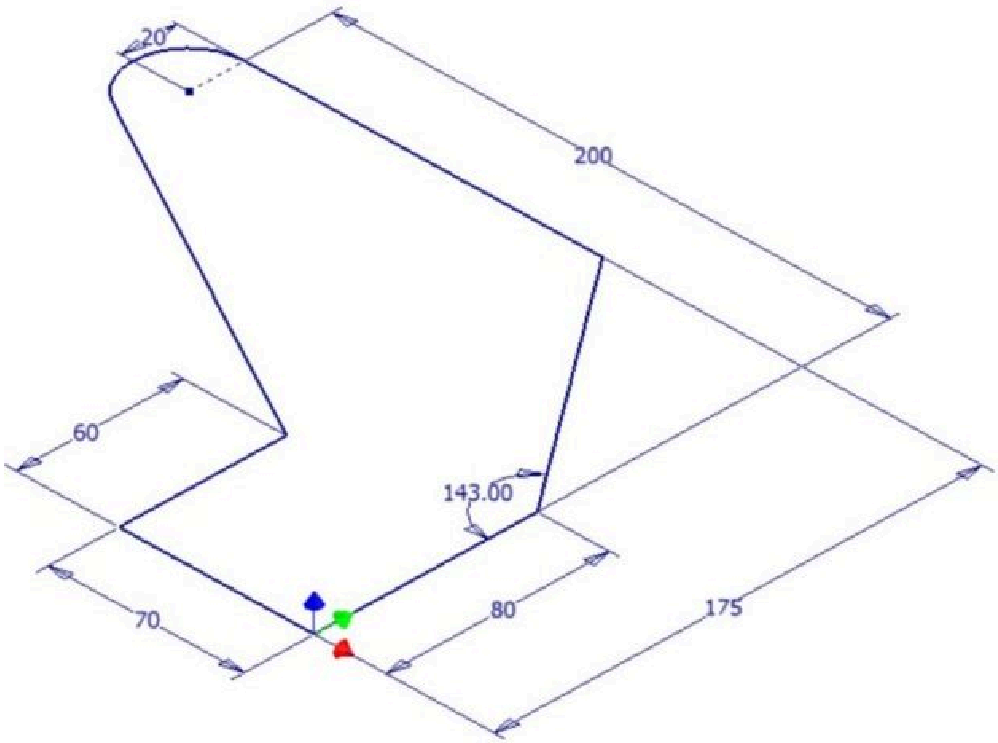


Figure Step 5
Editing the Base Sketch [Click to see image full size]

Step 6

Edit the feature Extrusion1 and change the thickness of the base sketch from 15mm to 20 mm. Return to model mode.

Step 7

Using the next five figures below, change the colors of the machined faces shown to: Aluminum (Polished). (Figure Step 7A, 7B, 7C, 7D, and 7E)

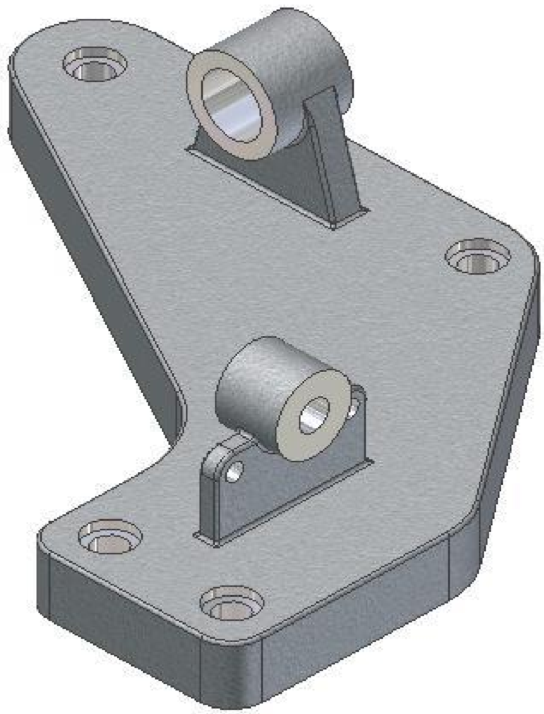


Figure Step 7A
Solid Model – Colored Faces

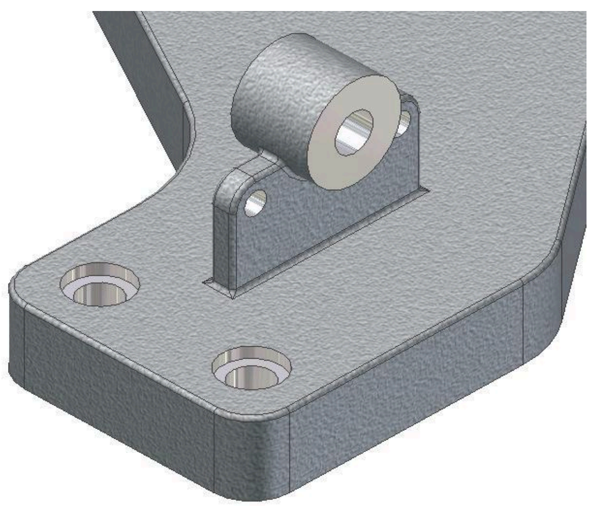


Figure Step 7B
Solid Model – Colored Faces

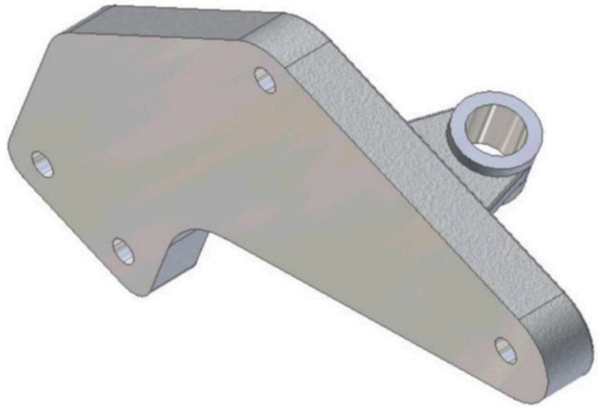


Figure Step 7C
Solid Model – Colored Faces

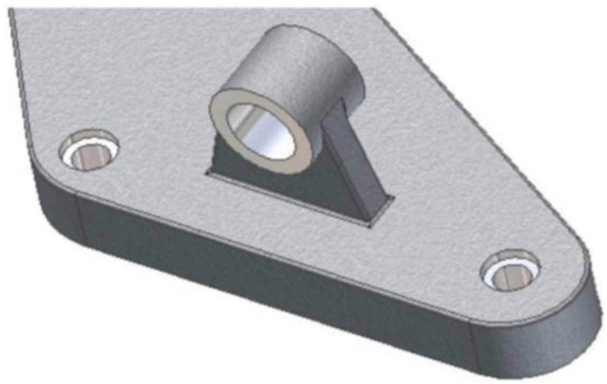


Figure Step 7D
Solid Model – Colored Faces

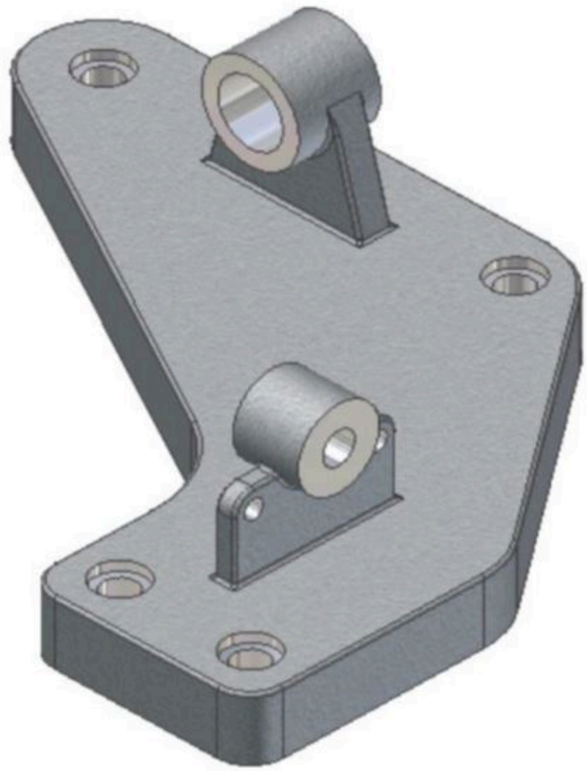


Figure Step 7E
Solid Model – Colored Faces

Lab Exercise 20-2

Time allowed: 60 minutes.

Part Name: Inventor Lab 20-2	**Project**: Inventor Course	**Units**: Inches
Template: N/A	**Color**: Blue Gray (Light)	**Material**: Cast Iron

Step 1

Open part: Inventor Lab 18-2.ipt that you created in Module 18. (Figure Step 1A and 1B)

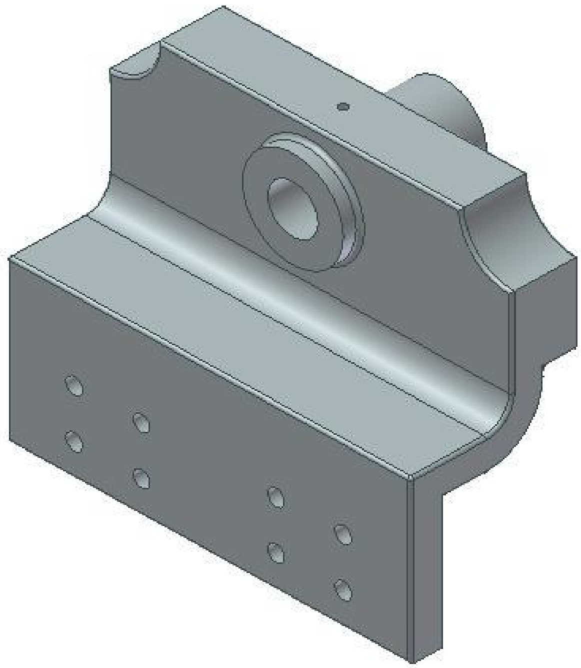


Figure Step 1A
Solid Model – Home View

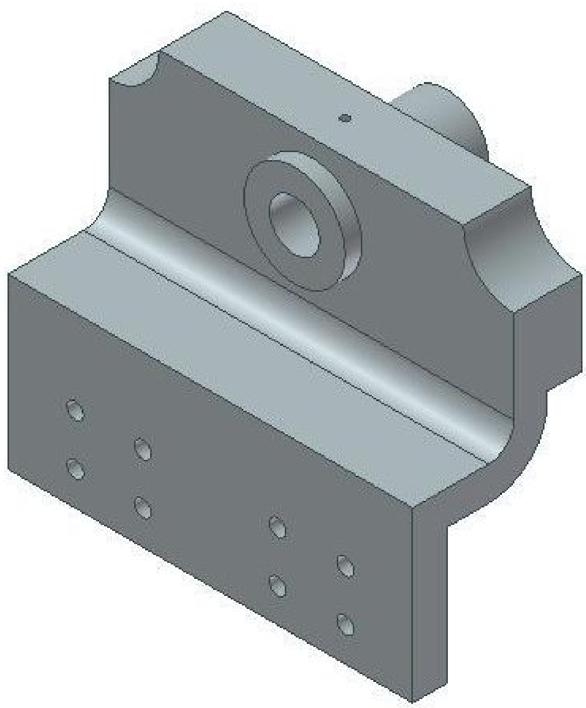


Figure Step 1B
Solid Model – Without the Small Fillets

Step 2

Save the file as: Inventor Lab 20-2.

Step 3

Change the colour to: Blue Gray (Light)

Step 4

Move the End of Part Icon up above the small fillets as shown below. (Figure Step 4)

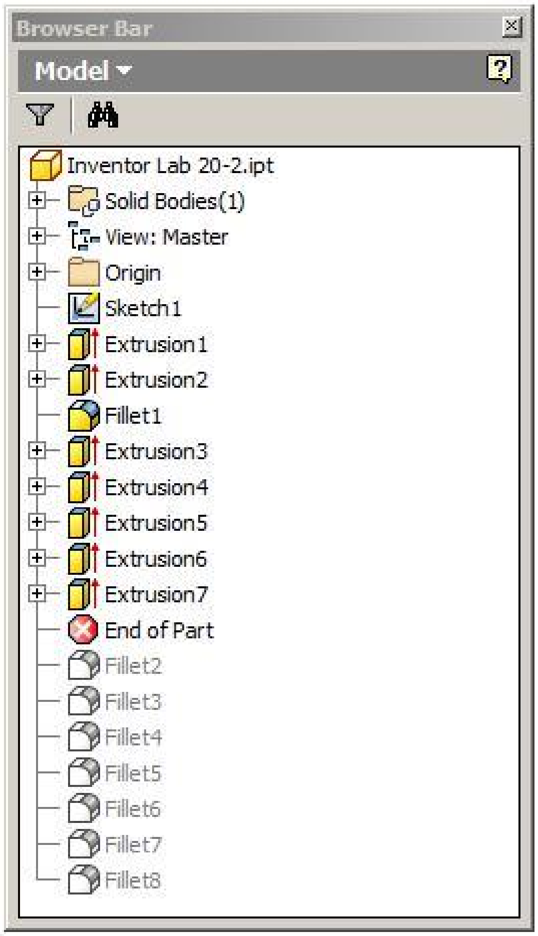


Figure Step 4

Step 5

Find the following with a precision of 5 decimal points. (Figure Step 5)

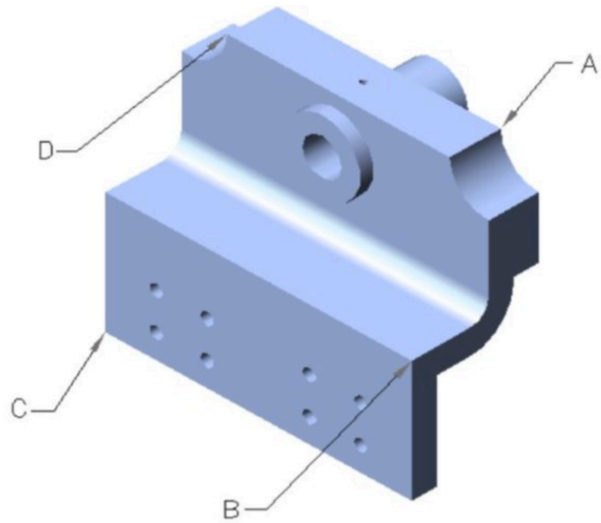


Figure Step 5

A The XYZ coordinates of corner A.

____

B The distance from corner B to corner C.

____

C The distance from corner B to corner D.

____

D The area of the shaded area the figure. Do NOT include the area of the cylinder. (Figure Step 5D)

____

E The perimeter of the shaded plane. (Figure Step 5E)

____

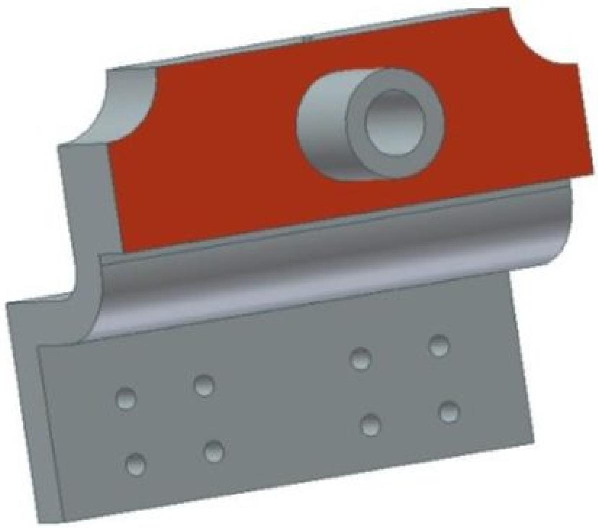


Figure Step 5D and 5E)

F The area of the shaded area of the figure WITHOUT the eight small holes. (Figure Step 5F) ____________________________

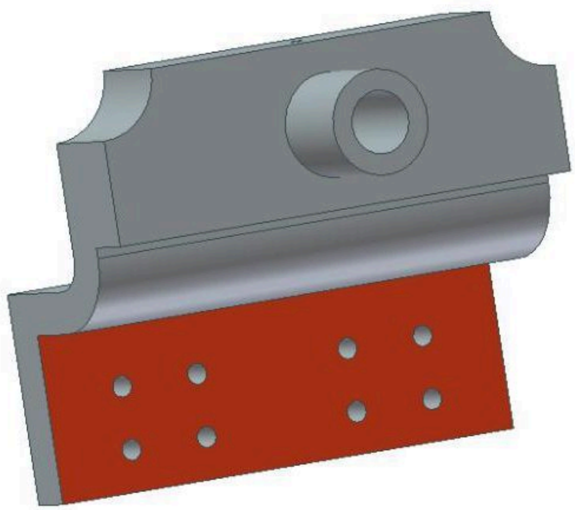


Figure Step 5F

Step 6

Move the End of Part icon back to bottom.

Step 7

Using the two figures, change the colors of the faces shown to: Blue Pastel. (Figure Step 7A and 7B)

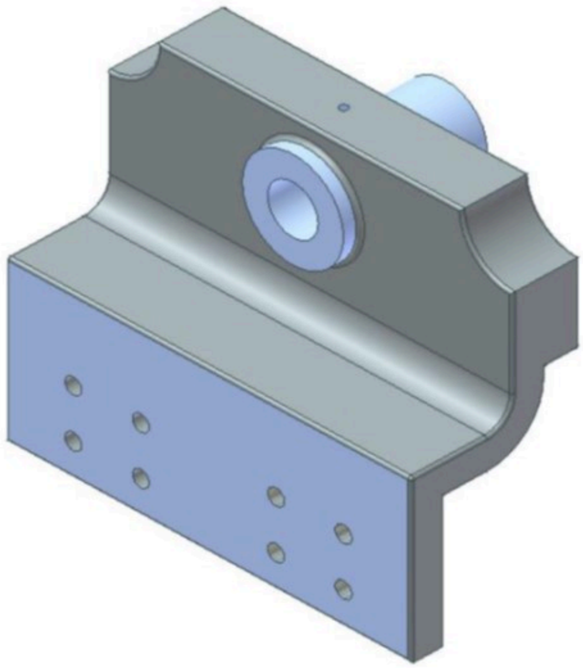


Figure Step 7A
Solid Model – Colored Faces

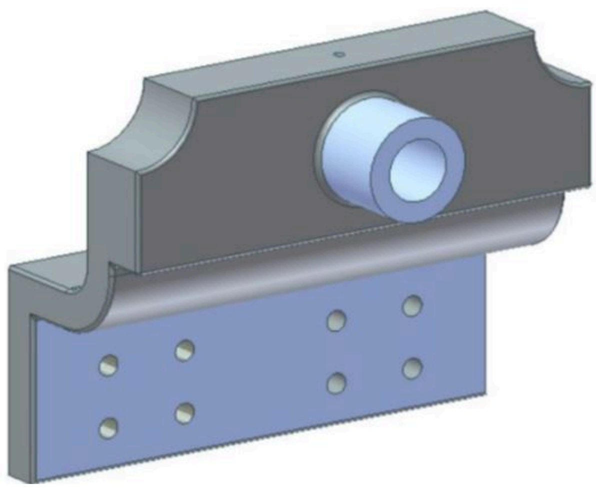


Figure Step 7B
Solid Model – Colored Faces

Step 8

Set the material to: Cast Iron and find the following:

A The mass in pounds.

B The mass in grams.

Step 9

Save and close the part.

Module 21 Competency Test No. 4 Open Book

Learning Outcomes

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

1. Within a two hour time limit, complete a written exam and a lab exercise.

The Inventor 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 you have completed this module.

Multiple Choice Questions

Select the BEST answer.

1. When you are selecting objects with a window, which window selects all the objects that are totally inside it and the ones that it crosses?
   A. Square Window
   B. Crossing Window
   C. Extruded Window
   D. Rectangular Window
   E. Polygon Window
2. What is the name for two or more objects that are connected at their endpoints and then treated as one object in the OFFSET command?
   A. Loop

   B. Continuous
   C. Area
   D. Polyloop
   E. Window

3. What are the basic sketching planes (XY, XZ, YZ), the three axis (X, Y, Z) and the Center Point called?
   A. Work Planes
   B. Work Points
   C. Work Features
   D. Work Sketches
   E. Work Axis
4. What is the maximum number of sides that the POLYGON command can draw a regular polygon?
   A. 20
   B. 100
   C. 120
   D. 180
   E. 256
5. When the graphic cursor appears as a ruler, what units are you measuring in? Select the BEST answer.
   A. English Units
   B. Document Units
   C. System Units
   D. Metric Units
   E. Actual Units
6. What command is used to extend the length of an existing line or an arc?
   A. LENGTHEN
   B. EXTEND
   C. LINE
   D. LENGTH
   E. STRETCH
7. What does an aligned dimension measure?
   A. The delta X or delta Y distance between two points.
   B. The diameter of circle or radius of an arc.

   C. The true length of a line or the true distance between two points.
   D. The horizontal or vertical distance between two points.
   E. The angle between two lines or the angle between the imaginary lines between three points.

8. What best describes the threads created using the THREAD command?
   A. Graphical representations of the actual thread.
   B. A texture
   C. The actual thread.
   D. They are enlarged to look better.
   E. Center

9. What is a parametric construction plane inserted on the model or in model space called?
   A. A Work Plane
   B. A Work Point
   C. A Work Feature
   D. A Work Sketch
   E. A Work Model

10. What is an angular dimension measuring?
    A. The delta X or delta Y distance between two points.
    B. The diameter of circle or radius of an arc.
    C. The true length of a line or the true distance between two points.
    D. The horizontal or vertical distance between two points.
    E. The angle between two lines or the angle between the imaginary lines between three points.

Lab Exercise 21-1

Time allowed: 2 hours.

Part Name	Project	Units	Template	Color	Material
Inventor Lab 21-1	Inventor Course	Millimeters	Metric-Modules Part (mm).ipt	Machined-Aluminum	Aluminum-6061

Step 1

Project the Center Point onto the Base plane.

Step 2

Note the location of X0Y0Z0. Draw the necessary sketches and extrude them to produce the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to maintain the objects shape and size. All sketches must be fully constrained. (Figure Step 2A and 2B)

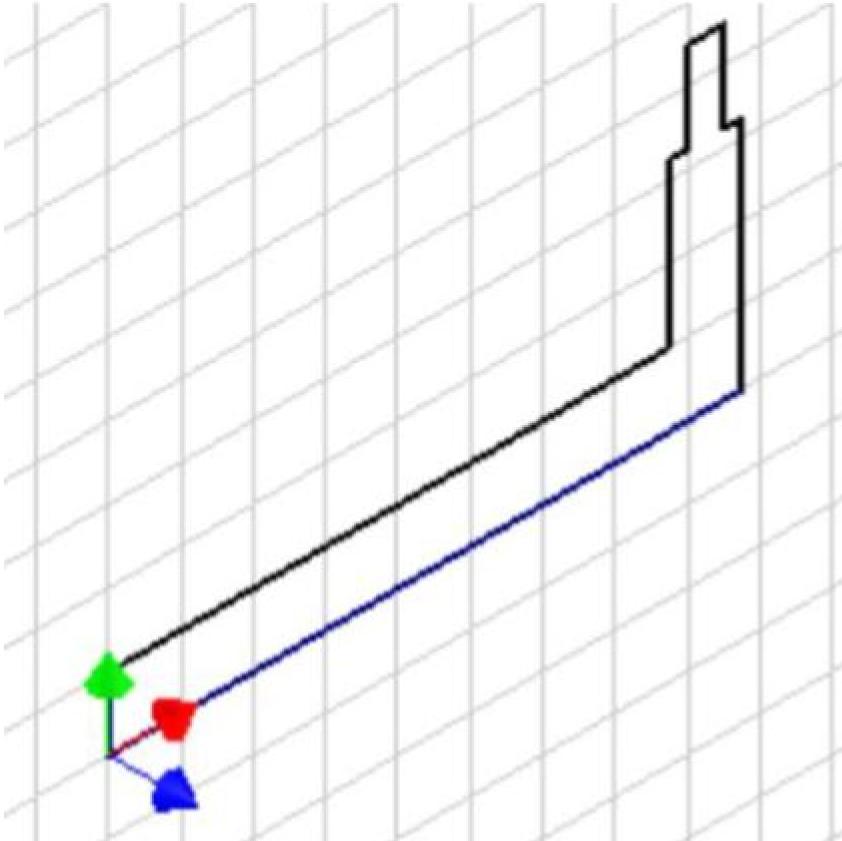


Figure Step 2A
Suggested Base Sketch –
Right Side (YZ) Plane

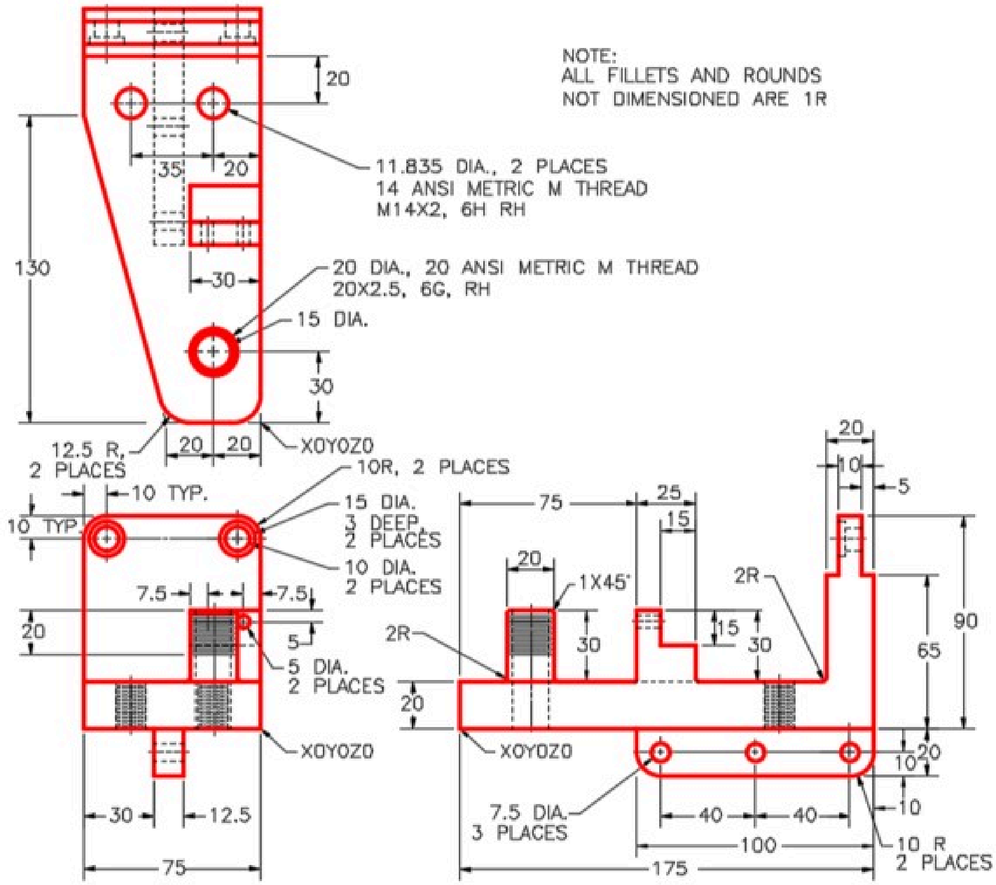


Figure Step 2B
Dimensioned Multiview Drawing
Competency [Click to see image full size]

Step 3

Create the fillets and chamfers after the model is totally constructed.

Step 4

Apply the colour shown above. (Figure Step 4)

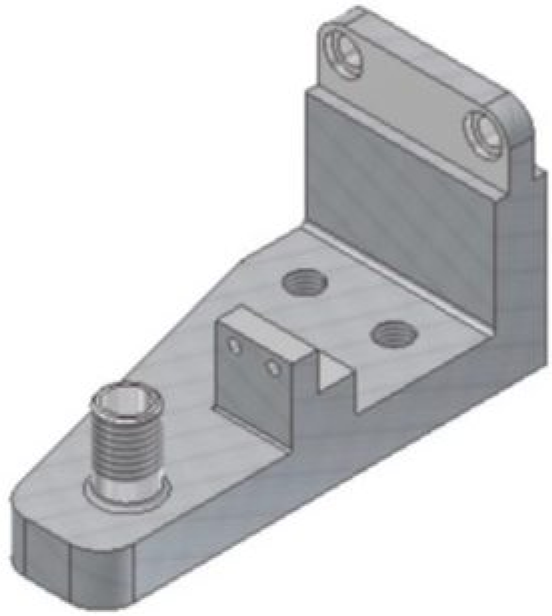


Figure Step 4
Solid Model
Home View

Step 5

Move the End of Part Icon up above the fillets as shown in the figure. Suppress all of the threaded features. (Figure Step 5)

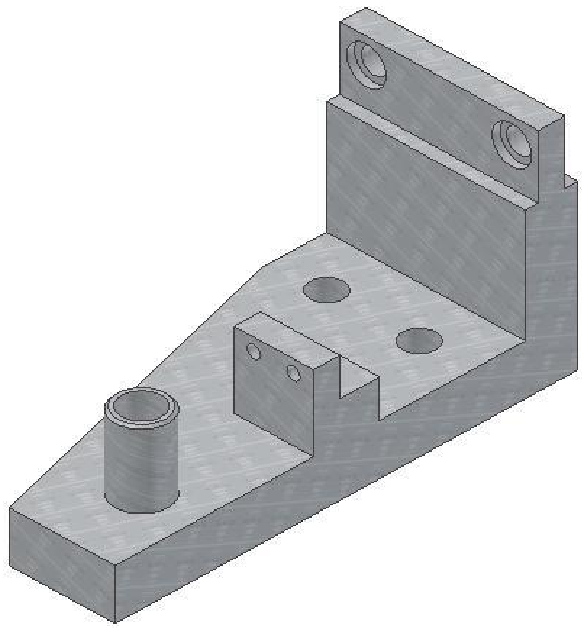


Figure Step 6

Step 6

Find the following to a precision of 6 decimal points: (Figure Step 6)

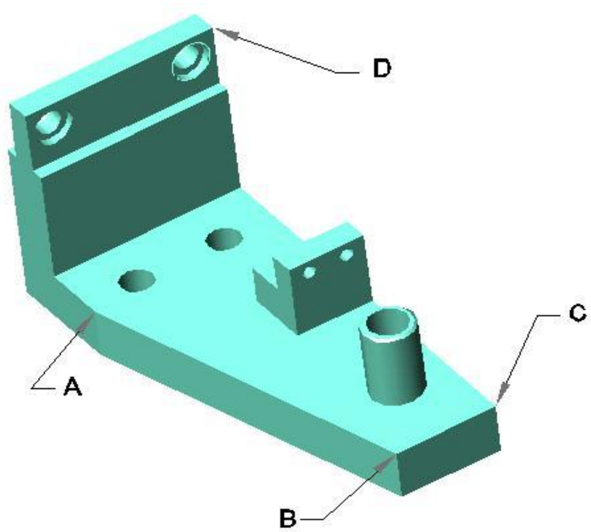


Figure Step 7

A The XYZ coordinates of corner D.

____

B The length of the edge from corner A to corner B.

____

C The distance from corner A to corner C.

____

D The distance from corner C to corner D.

____

E The angle between the edges B to C and B

____

Step 8

The area of the shaded area of Figure Step 8. Do NOT include the area of features on the plane.

____

Step 9

The perimeter of the shaded plane in Figure Step 9.

____

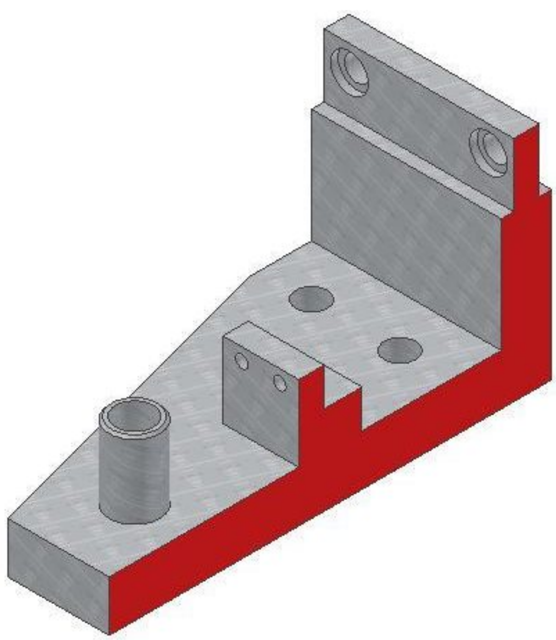


Figure Step 9

Step 10

The area of the shaded area of Figure Step 10. The area should include the complete surface without any features.

____

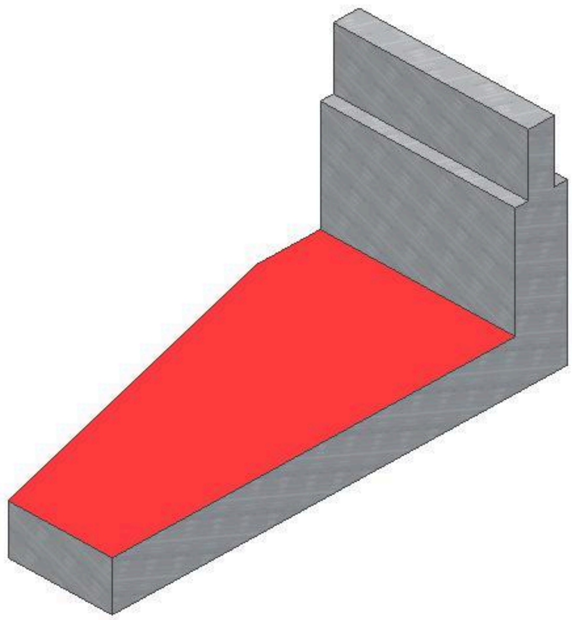


Figure Step 10

Step 11

Using the three figures below, change the colors of the faces shown to: Aluminum – Polished. (Figure Step 11A, 11B, and 11C)

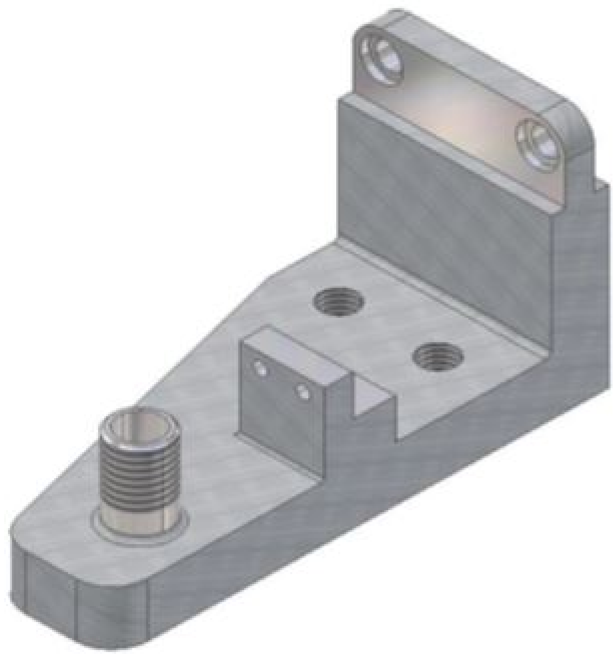


Figure Step 11A

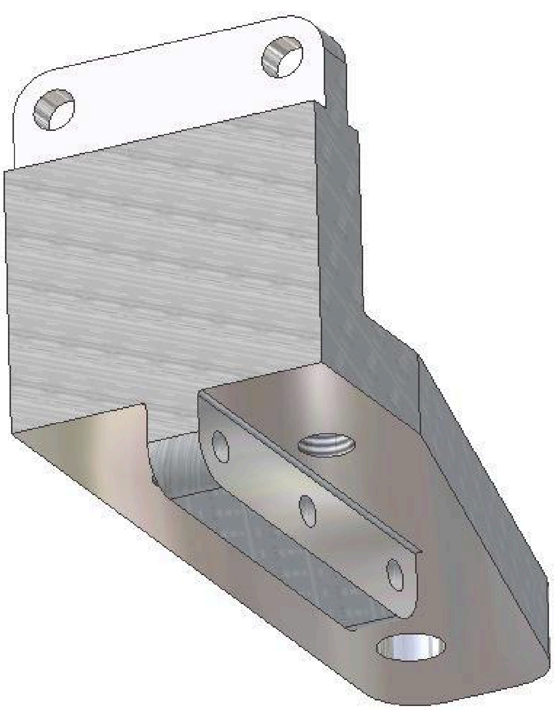


Figure Step 11B

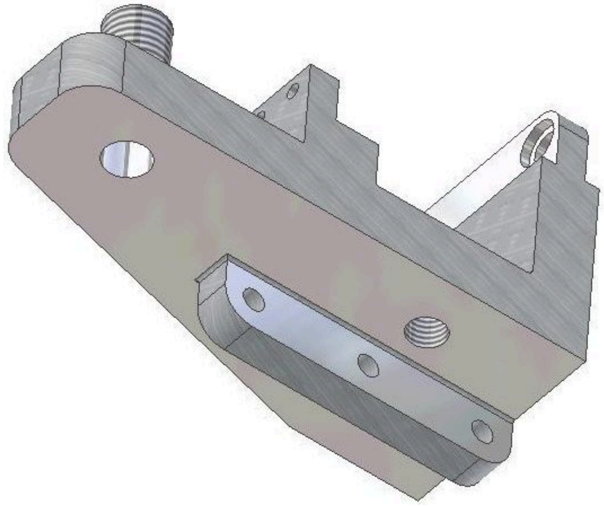


Figure Step 11C

Step 12

- If you are a student completing this course in a classroom setting, your instructor will give you instructions on what to do after you complete this module.
- If you are an online student doing this course by correspondence, send an email to your instructor with the answers from the measurement questions in this module. For example:

7A	The XYZ coordinates of corner D is ____________ .
7B	The length of the edge from corner A to corner B is ____________ .
7C	The distance from corner A to corner C is ____________ .
7D	The distance from corner C to corner D is ____________ .
7E	The angle between the edges B to C and B to A is ____________ .
8	The area of the shaded area of Figure Step 8 is ____________ .
9	The perimeter of the shaded plane in Figure Step 9 is ____________ .
10	The area of the shaded area of Figure Step 10 is ____________ .
12A	The mass of the solid model in grams is ____________ .
12B	The mass of the solid model in pounds is ____________ .

The answers in the email can be given as follows:

7A	____________ .
7B	____________ .
7C	____________ .
7D	____________ .
7E	____________ .
8	____________ .
9	____________ .
10	____________ .
12A	____________ .
12B	____________ .

Part 5

Module 22 Assemblies

Learning Outcomes

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

1. Describe an assembly and explain the difference between a top-down and a bottom-up assembly.
2. Describe and apply the SLICE GRAPHICS command.
3. Describe the PLACE COMPONENT and PLACE CONSTRAINT commands and apply them to assemble a series of parts to create an assembled model.

An Assembly File

An *assembly* file contains the information required to assemble two or more part files to create an assembled model. See Figure 22-1. As the model is assembled, assembly constraints must be assigned so that each part knows how it aligns or fits with the other parts in the assembled model. A part (.ipt) file that has been placed in the assembly can be edited while in the assembly file and the modifications will be saved back to the original part file. On the other hand, if the original part file is modified after the assembly file has being created, the modifications will automatically display in the assembly. An assembly file does not actually contain any of the part files that are placed in the file, it simply contains a reference to them.

Figure 22-1
An Assembled Model

A *reference* is a link back to the part files. The part files that have been placed in an assembly file must be available to Inventor to display them when an assembly file is opened. The current project file keeps track of those links and will automatically keep track of the location of all the part files. If an assembly file is sent to a client or an associate, the part files that were placed in the assembly file must also be included.

Creating an Assembly File

To create an assembly file, use the NEW command and select an assembly template rather than a part template. An assembly file has the extension .iam. IAM is an acronym for Inventor Assembly.

MUST KNOW: An assembly file contains the information required to assemble two or more part files to create the assembled model. An assembly file has the file extension .iam. IAM is an acronym for Inventor Assembly.

Bottom-up vs Top-down

Assembly files can be created using either the bottom-up or the top-down method. A *bottom-up* assembly is an assembly created from a series of part files that were previously created and saved in their own .ipt file. A *top-down* assembly is an assembly file where all parts of the assembly are created on the fly. In other word, they are created in the active assembly file, one at a time. They are aligned and constrained in their correct position in relationship to the other parts in the assembled model. Inventor will save each part in its own file with the extension .ipt. In the Inventor book, only the bottom-up method is taught.

The Browser Bar in an Assembly

When an assembly is the active file, the Browser bar will display all of the parts that have been placed in the assembly. If a part in the browser is expanded it will display its children which includes its alignment and constraints to the other parts of the assembled model. See Figure 22-2.

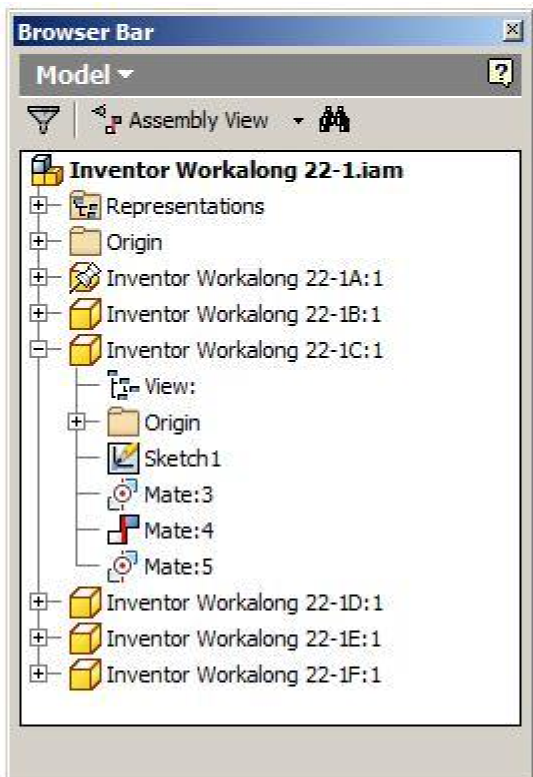


Figure 22-2
An Assemble in the Browser Bar Inventor

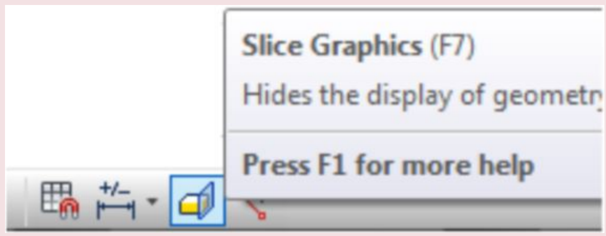
If the same part is placed more then once into the assembly file, the Browser bar will number them accordingly. For example, this may happen if an assembly required two identical bolts. Only one part file is created and named Bolt.ipt. It is then placed into the assembly file twice. The Browser bar would number the bolts parts as follows:

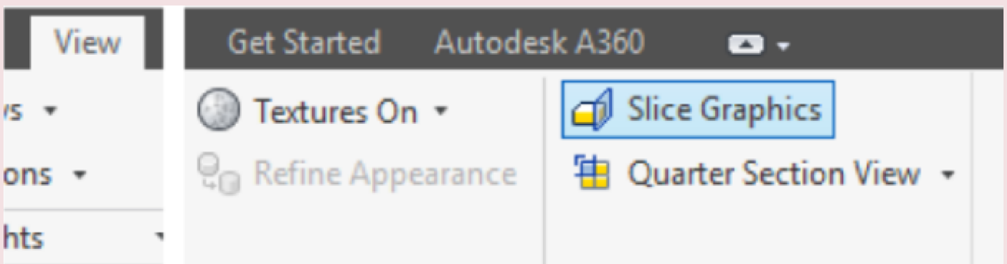
On the first occurrence: Bolt:1

On the second occurrence: Bolt:2

Inventor Command: SLICE GRAPHICS

The SLICE GRAPHICS command is used to slice away the model on the active sketching plane temporarily removing all of the material in front of the sketching plane. The sketching plane must be active before using this command.Shortcut: **F7**





WORK ALONG: Creating the Parts for an Assembly

Step 1

Check the default project and if necessary, set it to Inventor Course.

Step 2

Use the following instructions to complete all parts in this workalong. Create the following parts and ensure that you do the following:

A Each part must be saved in its own .ipt file.

B Project the Center Point onto the Base sketch plane and note the location of X0Y0Z0.

C Draw the necessary sketches and extrude or revolve them to produce the solid model shown. Apply all of the necessary geometrical and dimensional constraints to fully constrain each sketch.
D Apply the colour and material shown.

Step 3

Construct Part A. (Figure Step 3A, 3B, and 3C)

Part: Base

Part Name: Inventor Workalong 22-1A

Template: English-Modules Part (in).ipt

Color: Stainless – Brushed

Material: Stainless Steel

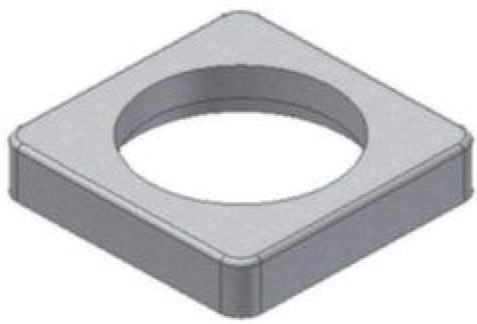


Figure Step 3A
Solid Model – Home View

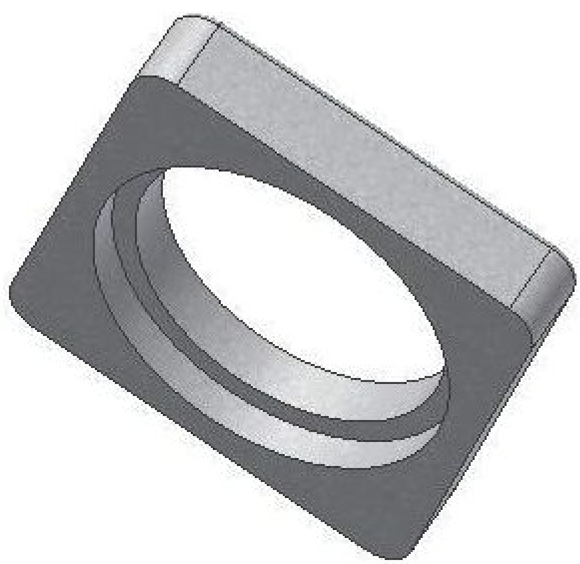


Figure Step 3B
Solid Model – Orbited View

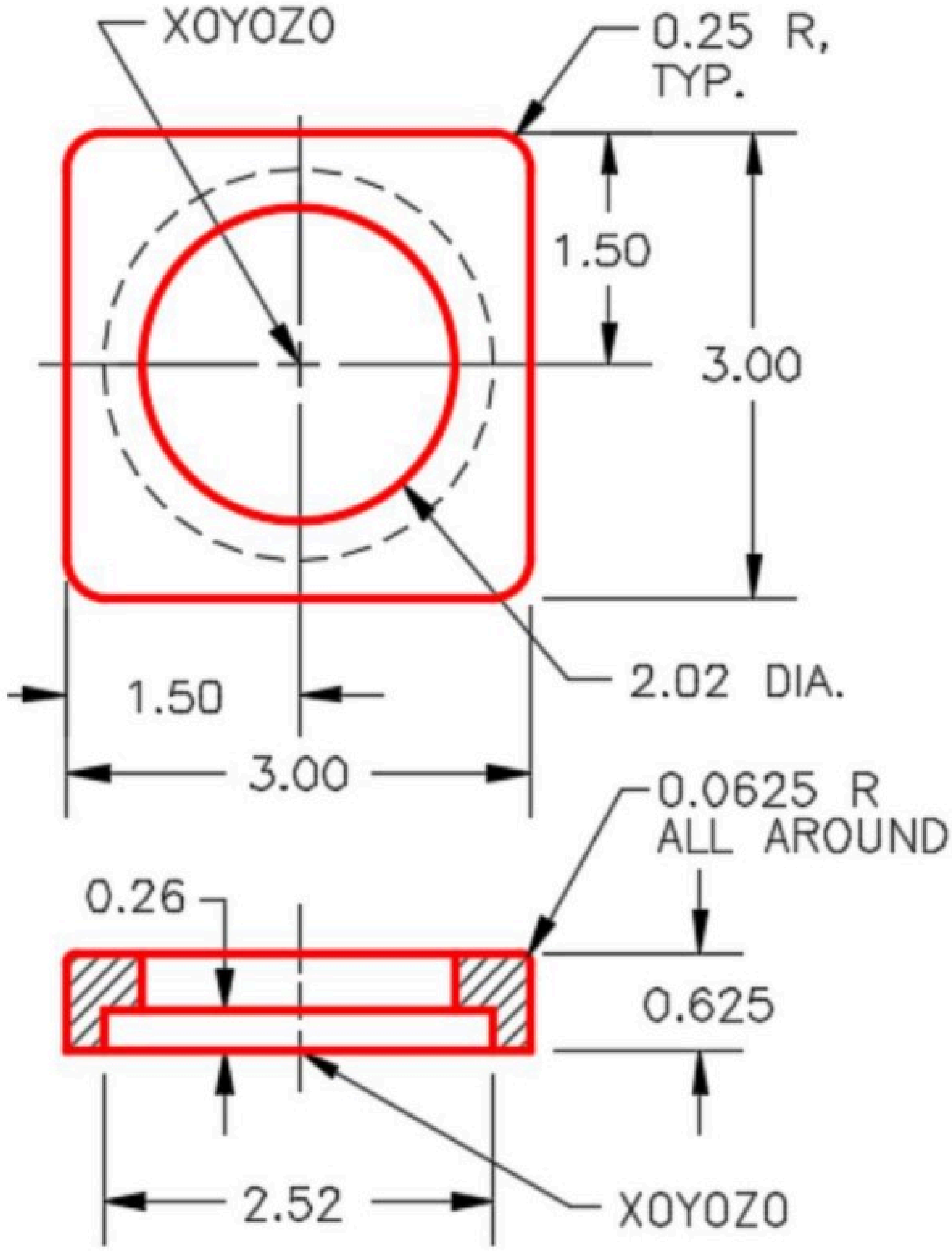


Figure Step 3C
Dimensioned Multiview Drawing [Click to see image full size]

Step 4

Construct Part B. (Figure Step 4A and 4B)

Note: See Steps 3 to 14 if you require help creating this model.

Part: Post

Part Name: Inventor Workalong 22-1B

Template: English-Modules Part (in).ipt

Color: Chrome – Black Polished

Material: Steel

Figure Step 4A Solid Model – Home View

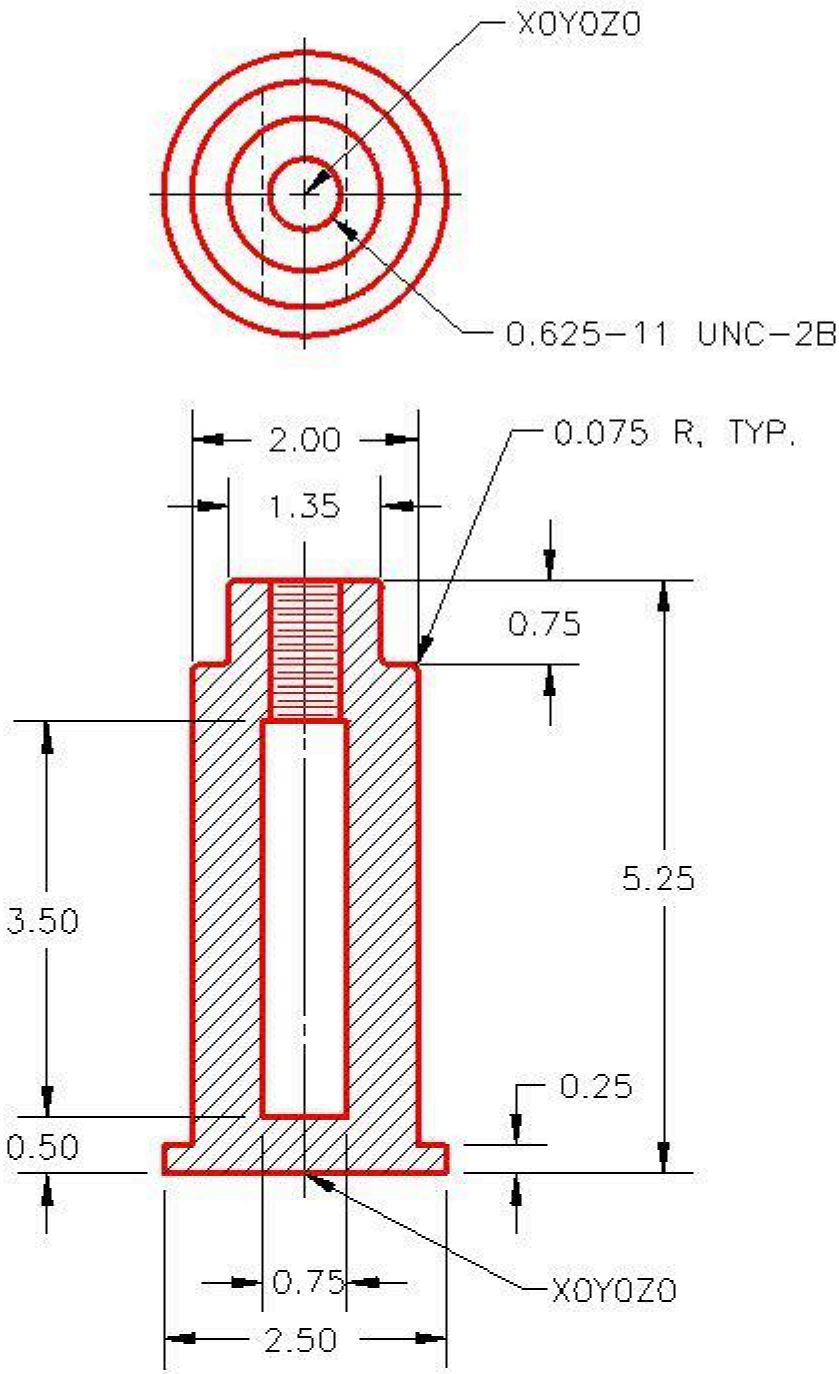


Figure Step 4B
Dimensioned Multiview Drawing

Step 5

Draw the Base model of part 22-1B by extruding circles. Your Base model should appear as shown in the figure. (Figure Step 5)

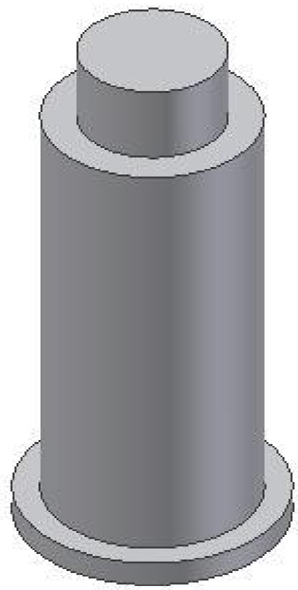


Figure Step 5

Step 6

Start a new sketch on the top of the model and draw a construction line from the centre to the edge of the top plane along the Y axis as shown in the figure. Hint: Ensure that you draw the line along the Y axis by snapping to the centre of the circle and to the grid on the Y axis. The length of the line is not important. (Figure Step 6)

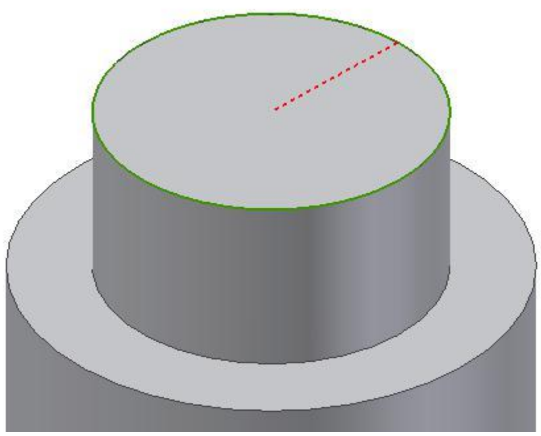


Figure Step 6

Step 7

Insert a Work Plane in the centre of the model using the Perpendicular to a Line method. Enter the WORK PLANE command and move the cursor to the end of the construction line. The Work Plane icon will display as shown in the figure. Click the end of the line when the icon displays. (Figure Step 7)

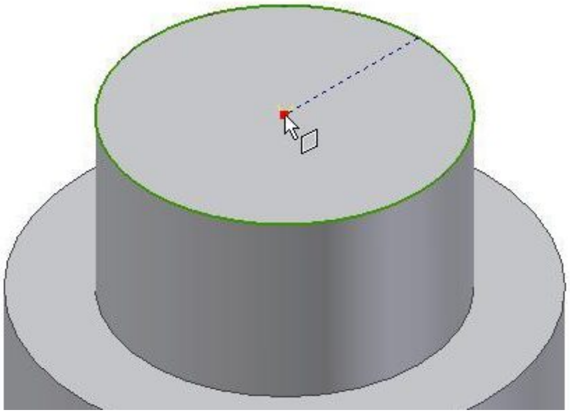


Figure Step 7

USER TIP: Before drawing each part, the orientation of the part when it is placed in the future assembly, should be considered. Doing that, will save a lot of time when creating the assembly model since it can be placed without any manipulation.

Step 8

Move the cursor onto the Y axis and it will display as shown in the figure. When the Y axis is displayed, click it and the Work Plane will display as shown. (Figure Step 8A and 8B)

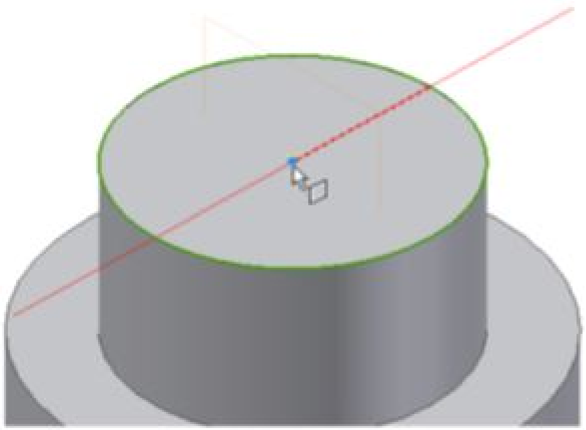


Figure Step 8A

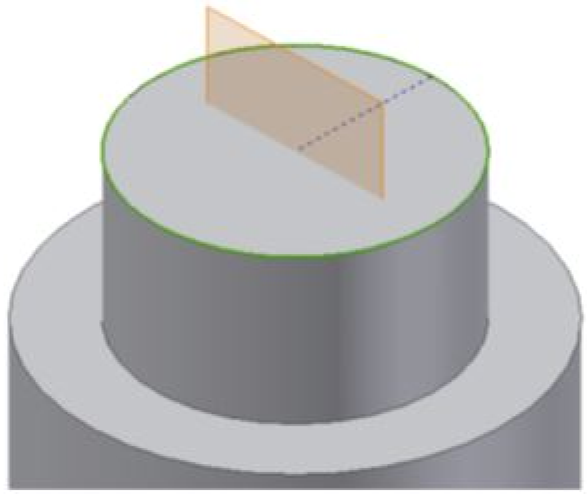


Figure Step 8B

Step 9

Enlarge the Work Plane to extend it past the edges of the model. One way to do this is to use the LOOK AT/VIEW FACE command and change the view so that it is looking perpendicular to the plane as shown in the figure. (Figure Step 9)

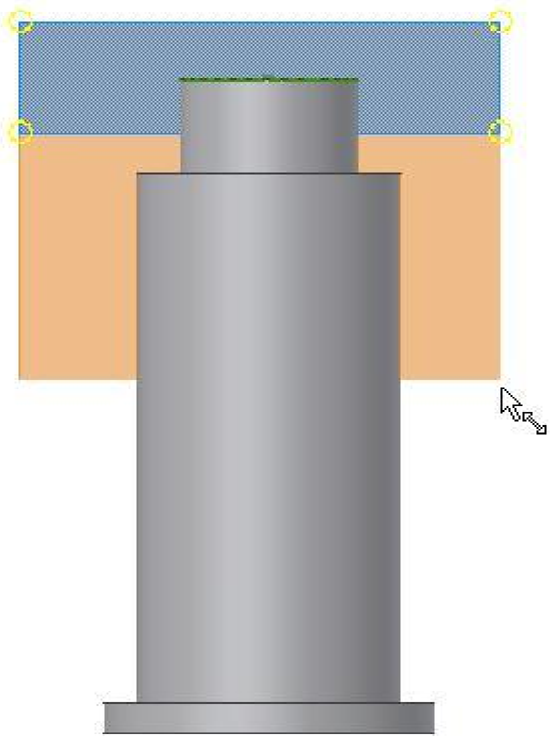


Figure Step 9

Step 10

Start a new sketch on the Work Plane. (Figure 10)

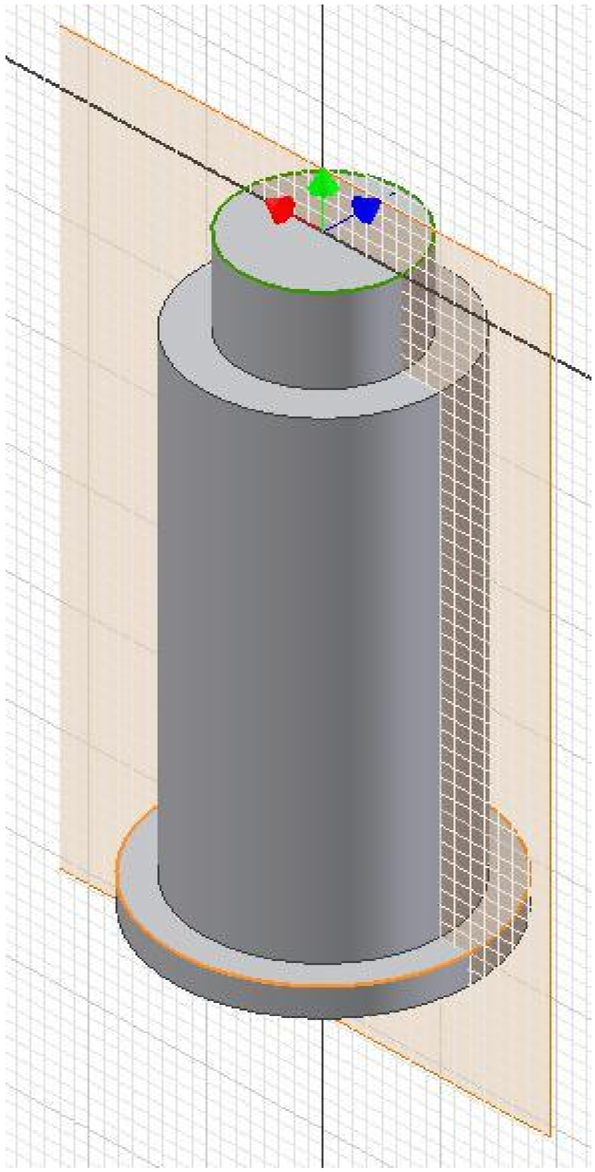


Figure Step 10

Step 11

Enter the SLICE GRAPHIC command (F7) and the model will display from the sketching plane back. Disable the grid display. It is easier to draw this sketch without the grid. (Figure Step11)

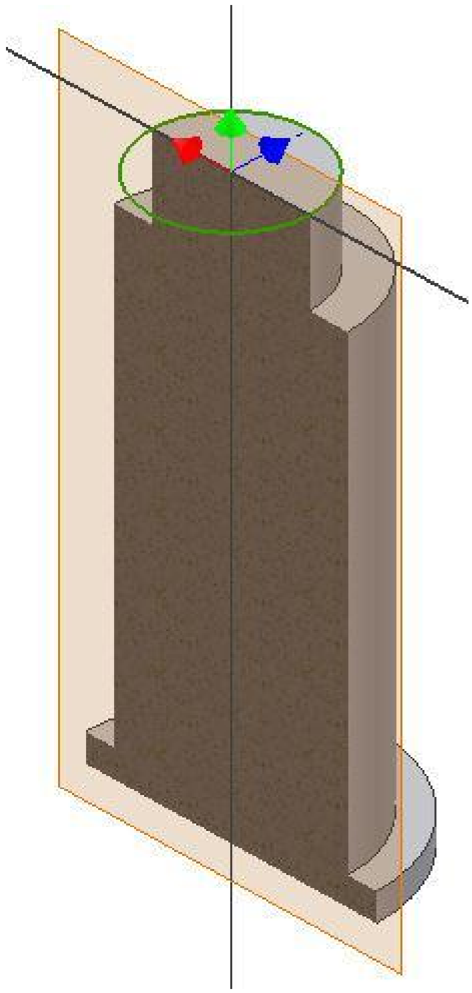


Figure Step 11

USER TIP: When enlarging the work plane, the cursor must display as the Stretch Move Stretch Cursor (two arrows) as shown. If you Cursor Cursor have trouble changing the cursor from the Move Cursor, zoom in closer and move the cursor onto one of the corners of the plane.

Step 12

Using the PROJECT GEOMETRY command, project the Z axis and the X axis onto the sketching plane. (Figure Step 12A and 12B)

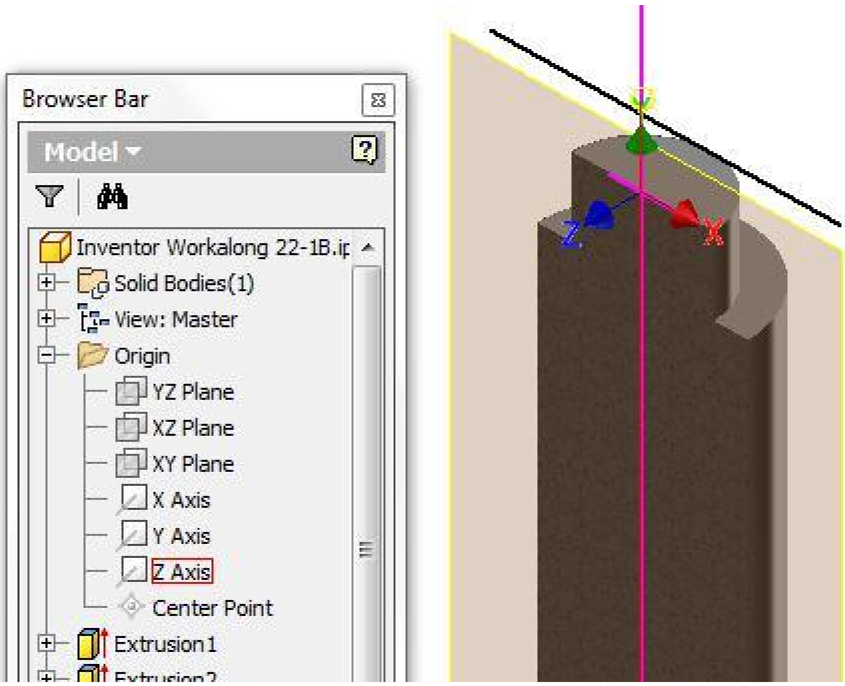


Figure Step 12A

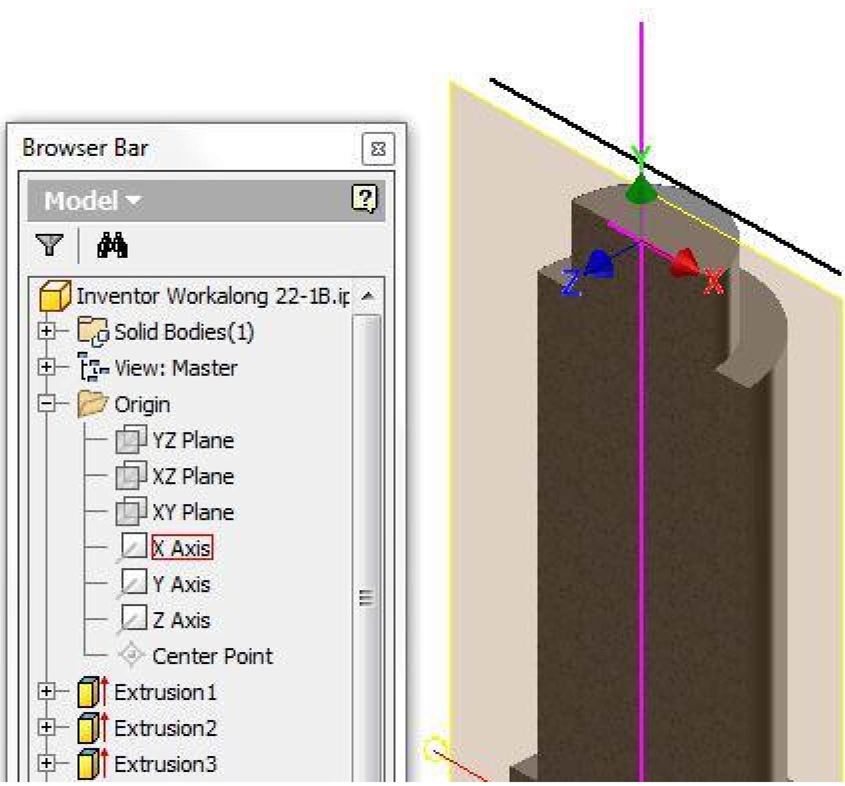


Figure Step 12B

Step 13

Using the OFFSET command, construct offsets to both the Z and X axis to start the construction of the slot. Trim the lines in the sketch and insert the dimensions to fully constrain the sketch. Ensure that you dimension from both sides of the Z axis as shown in the figure. (Figure Step 13A and 13B)

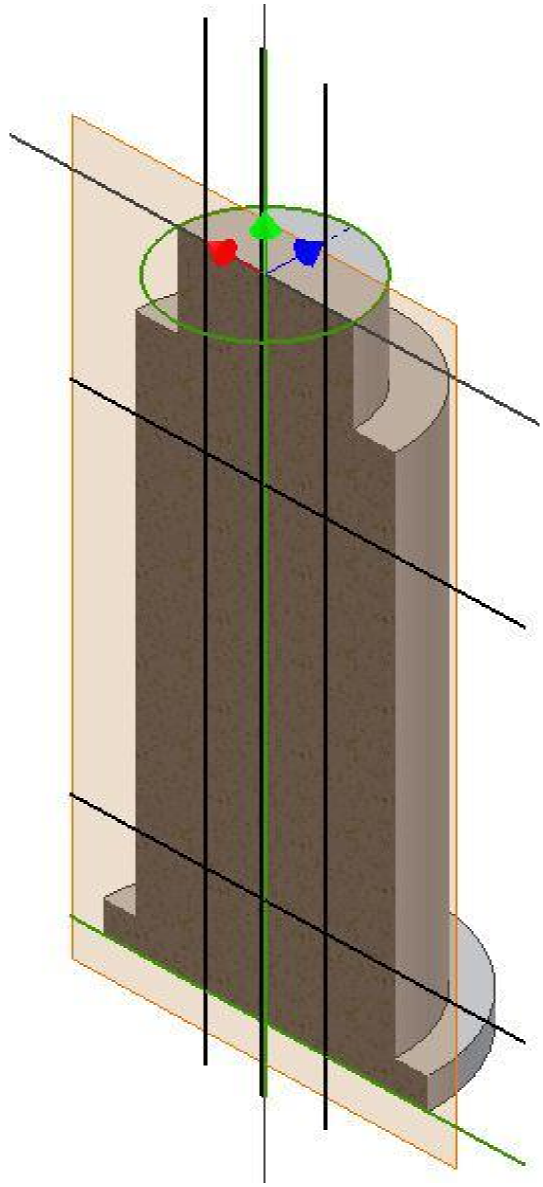


Figure Step 13A

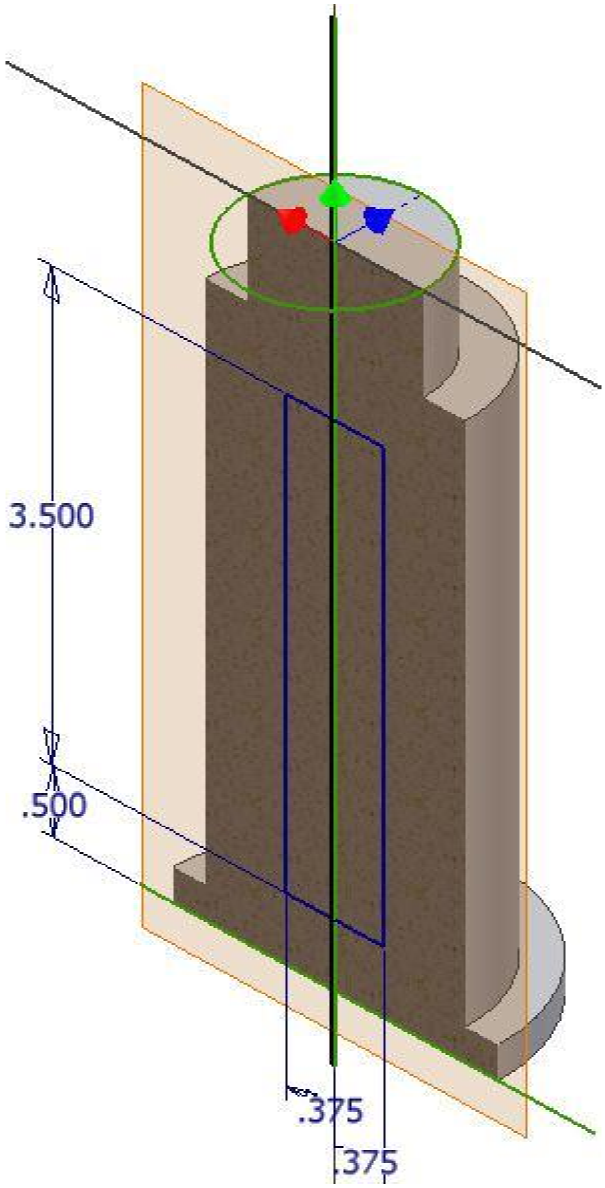


Figure Step 13B

MUST KNOW: An assembly file does do not contain any of the part files that are placed in the assembled model. It simply contains a reference to the part files. A reference is a link back to the part files. The part files that have been placed in an assembly file must be available to Inventor to display them when the assembly file is opened.
The project file keeps track of those links and will automatically keep track of the location of all the files. If an assembly file (.iam) is sent to a client or an associate, the part files (.ipt) that were placed in the assembly file must also be included otherwise they will not display when the file is opened.

Step 14

Extrude the sketch. Set it to extrude in both directions and to cut. Set the distance to All. Complete the part by adding the hole, the threads, the fillets, the colour, and the material as specified in Figure 14A, 14B, and 14C.

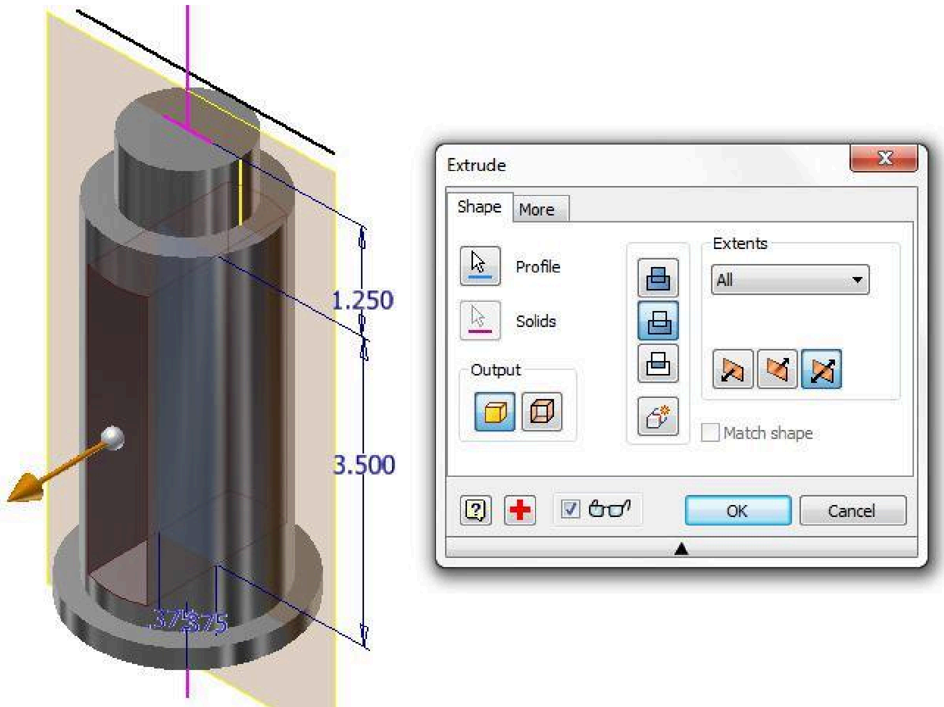


Figure Step 14A [Click to see image full size]

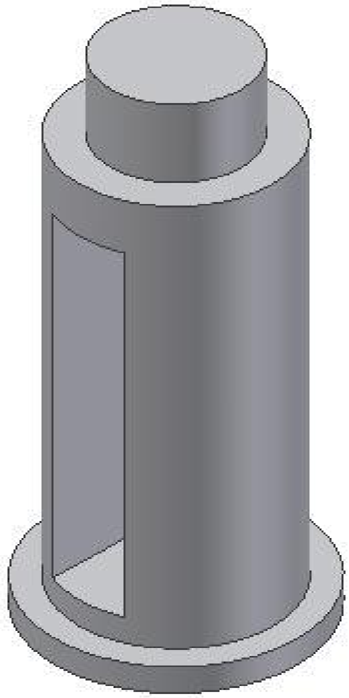


Figure 14B

Figure 14C

Step 15

Create Part C. (Figure Step 15A, 15B, 15C, and 15D):

Part: Wedge Ring

Part Name: Inventor Workalong 22-1C
Template: English-Modules Part (in).ipt
Color: Titanium – Polished
Material: Titanium

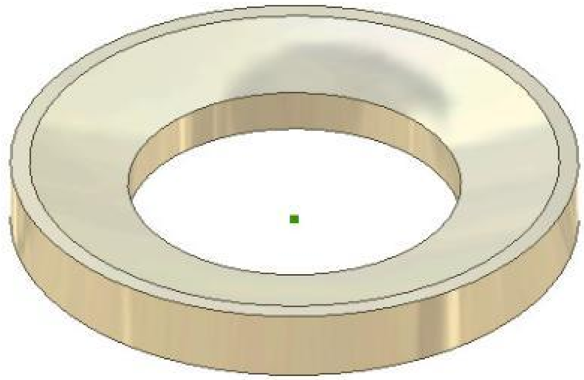


Figure Step 15A
Solid Model – Home View

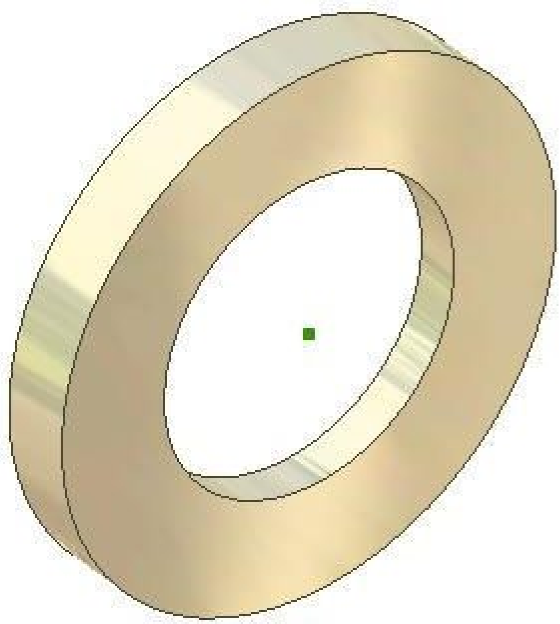


Figure Step 15B
Solid Model
– Orbited View

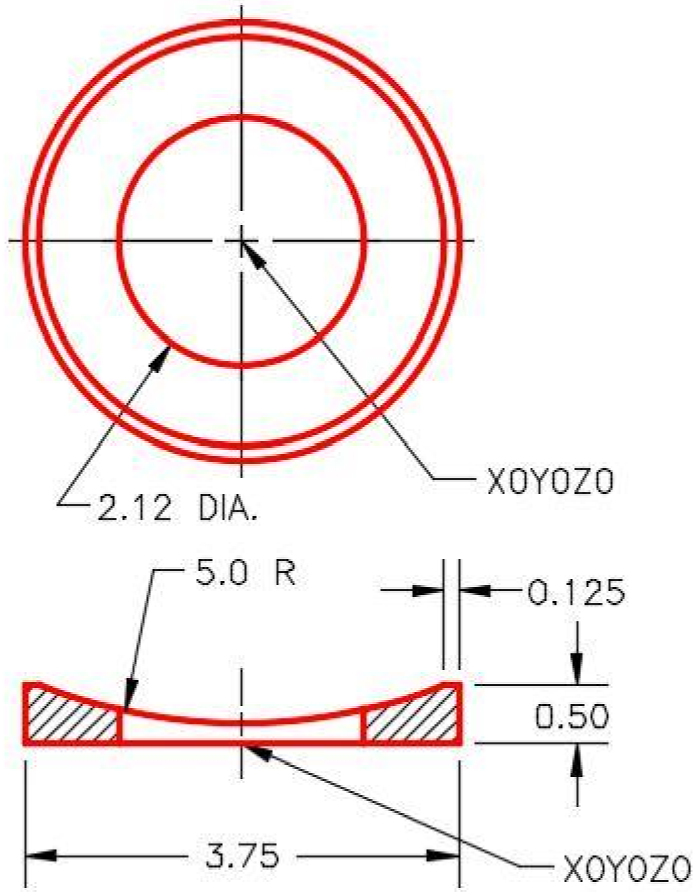


Figure Step 15C
Dimensioned Multiview Drawing

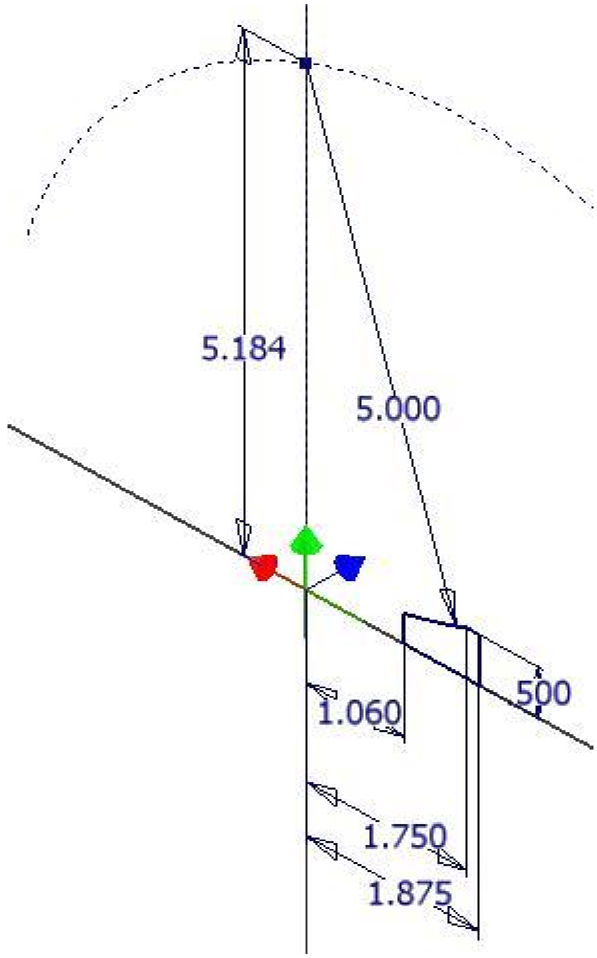


Figure Step 15D
The Base Sketch

AUTHOR'S COMMENTS: If you project the X and Z axis onto the plane, the same way you did in Step 12 and then construct arcs and offset lines it will be much easier to construct the Base sketch for this part. The last step is to revolve the sketch to complete the solid model.

Step 16

Construct Part D. (Figure Step 16A and 16B)
Part: Wedge
Part Name: Inventor Workalong 22-1D
Template: English-Modules Part (in).ipt
Color: Chrome – Polished
Material: Steel

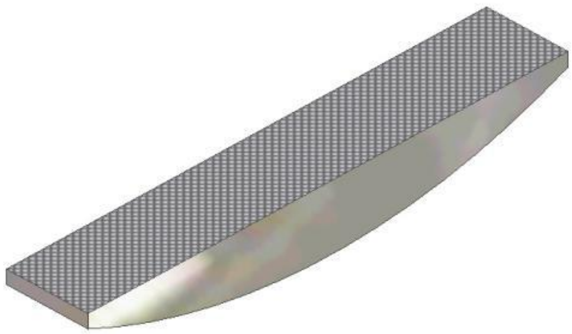


Figure Step 16A
Solid Model – Home View

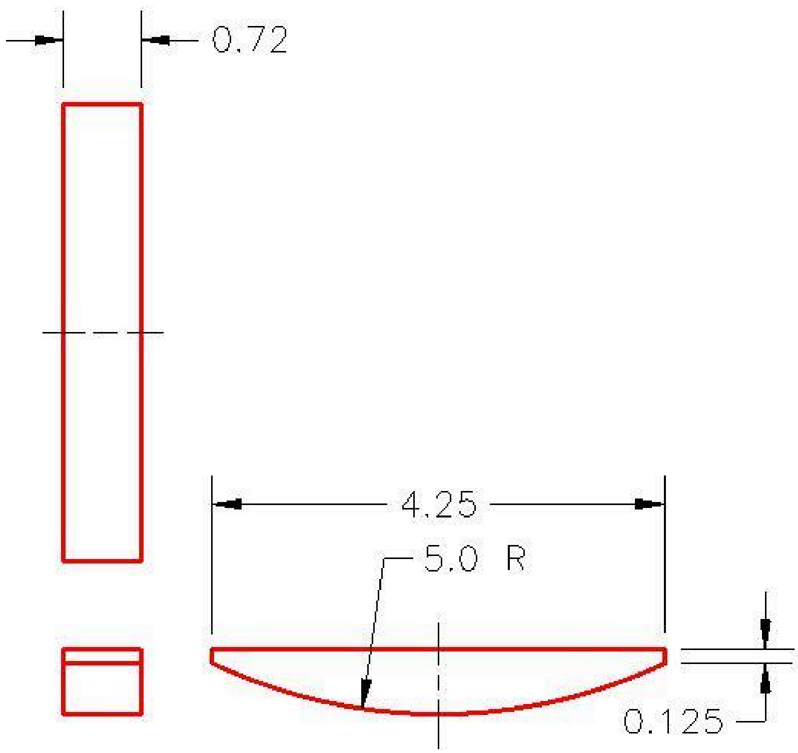


Figure Step 16B
Dimensioned Multiview Drawing [Click to see image full size]

Step 17

Change the face colour of the top face of the part to: Metal Steel (Knurled) as shown in Figure 16A.

Step 18

Construct Part F as follows: (Figure Step 18A and 18B)
Part: Handle
Part Name: Inventor Workalong 22-1E
Template: English-Modules Part (in).ipt
Color: Metal-AL-6061 – Machined
Material: Steel

Figure Step 18A
Solid Model – Home View

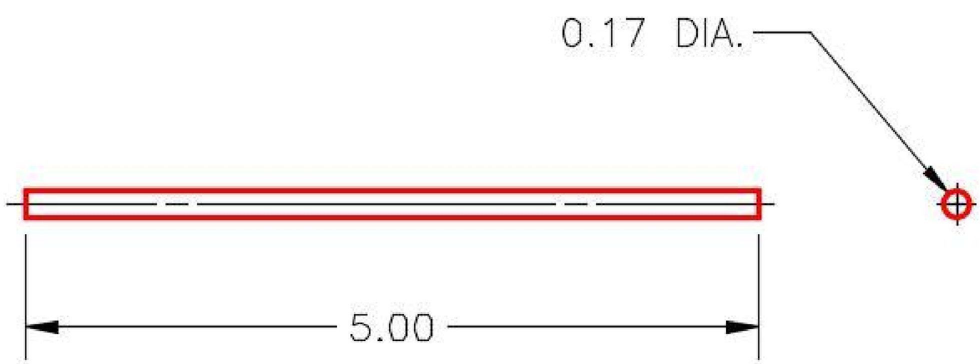


Figure Step 18B
Dimensioned Multiview Drawing

Step 19

Construct Part E as follows: (Figure Step 19A, 19B, and 19C)
Part: Screw
Part Name: Inventor Workalong 22-1F
Template: English-Modules Part (in).ipt
Color: Metal-AL-6061 – Machined
Material: Steel

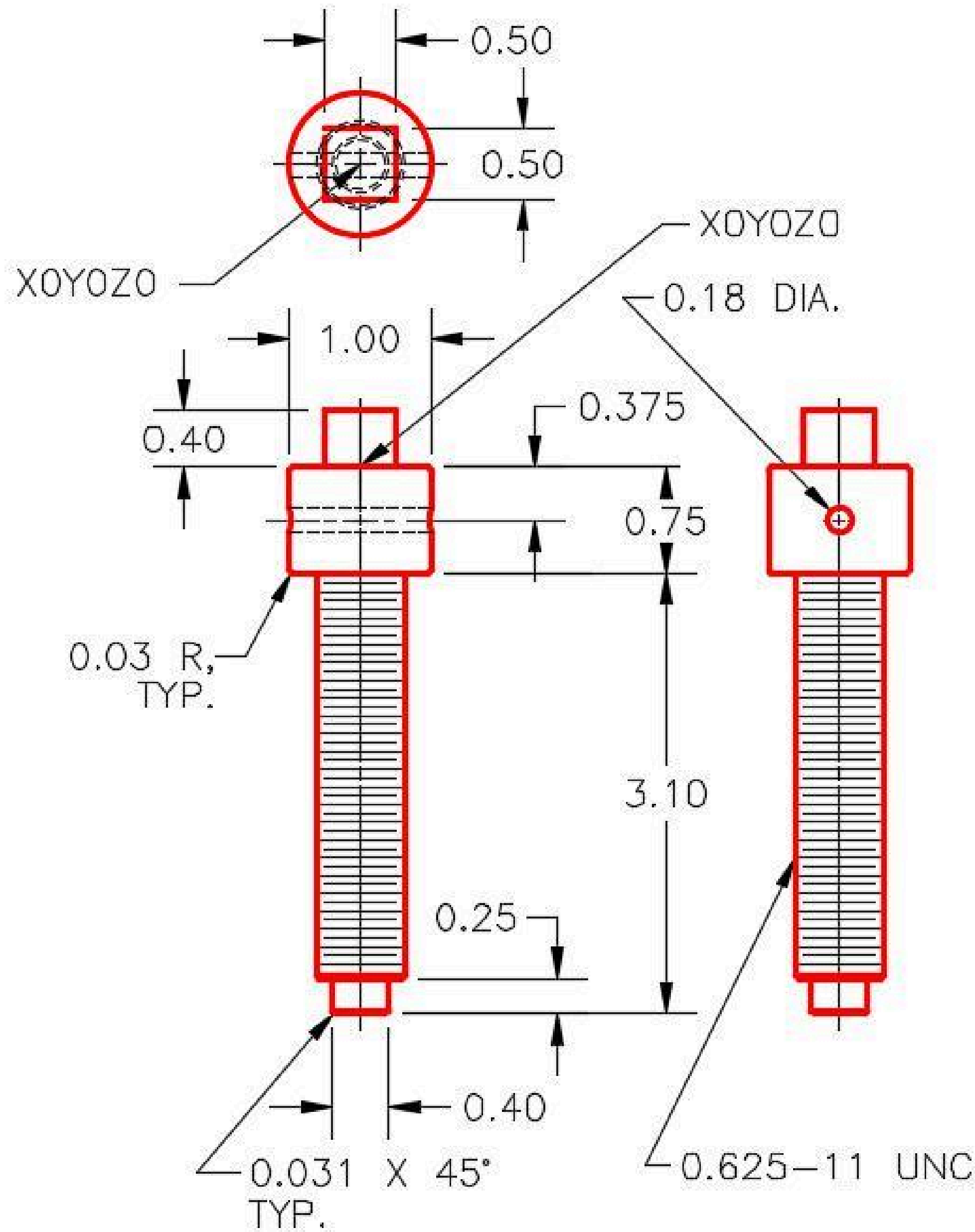


Figure Step 19A
Dimensioned Multiview Drawing [Click to see image full size]

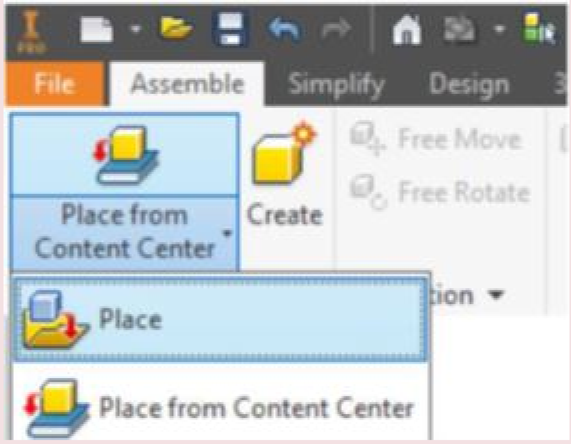
Figure Step 19B Solid Model – Home View

Figure Step 19C Solid Model – Orbited View

Step 20

Change the face colour of the cylindrical top of the part to: Metal Steel (Knurled) as shown in the solid model figure.

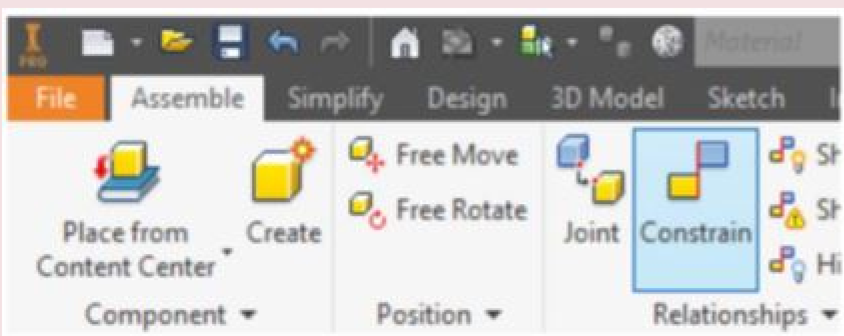
> Inventor Command: PLACE COMPONENT
>
> The PLACE COMPONENT command is used to insert a part or a component into an assembly file.
>
> Shortcut: **P**



Inventor Command: PLACE CONSTRAINT

The PLACE CONSTRAINT command is used to apply constraints of one part to another in the assembled model.

Shortcut: **C**



Grounded Parts

By default, the first part that is placed into an assembly file will be grounded. A *grounded* part is a part that has all of it degrees of freedom removed and is fully constrained in that file. It is important to ground at least one part of every assembly. If no parts are grounded, the assembly can be moved around in model space. Once one part is grounded, the other parts can be constrained to it making their movement relative to the grounded part. If required, more then one part can be grounded.

In the Browser bar, a grounded part will display a Push Pin icon as shown in Figure 22-3 and 22-4. To enable or disable a part being grounded, right-click the part in the Browser bar. In the Right-click menu, select Grounded. In Figure 22-5, the selected part is currently grounded, enabled and visible.

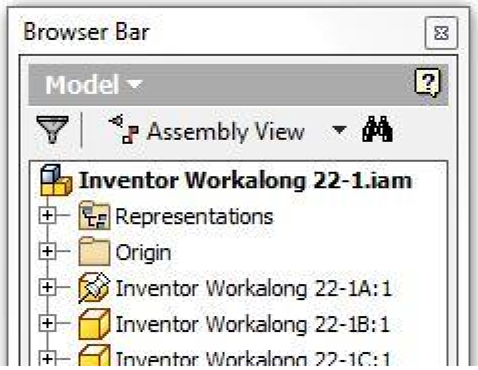


Figure 22-3 Grounded Part as it Displays in the Browser Bar

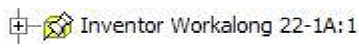


Figure 22-4 Grounded Part Icon

Part Visibility

The visibility of parts in an assembly can be enabled or disabled. When the part's visibility is disabled in the assembly, the part's icon in the Browser bar will display greyed out.

Enabled Parts

Parts can be enabled or disabled in the assembly file. If a part is enabled, it displays and can be selected in the assembly. If it is disabled, only an outline displays and It cannot be selected. Sometimes it is easier to disable some parts to make it easier to place additional parts.

Assembly Constrains

There are many different constraints used when creating an assembly. In this module, only the mate constraint will be taught. A *mate constraint* constrains two assembled parts to one another by mating their centerlines and/or by mating a face on one part to a face on the other part. Mating is the most common way to assemble two parts together. It often takes more then one constraint to assemble two parts together. Both parts must have a symmetrical feature to mate them using the centerline method.

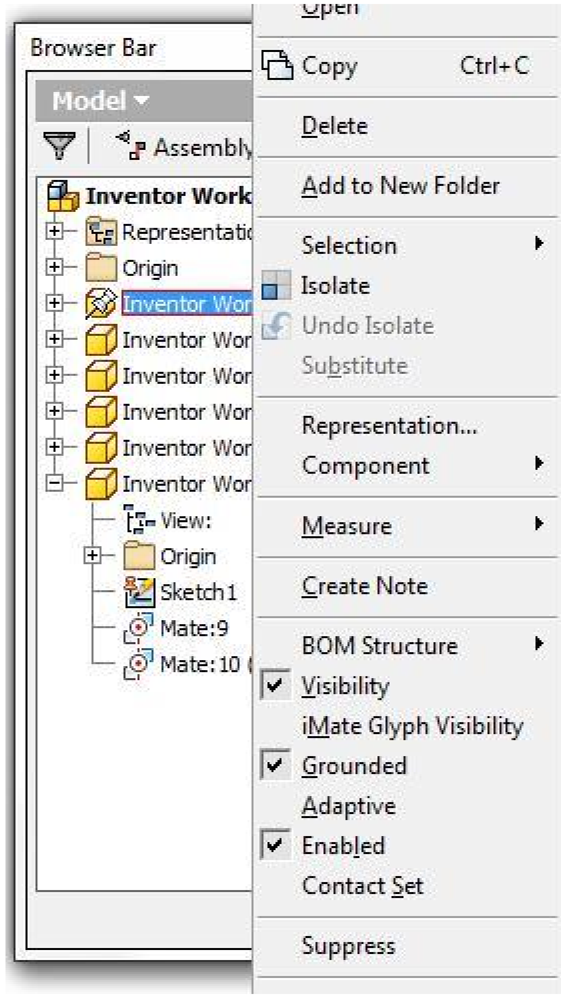


Figure 22-5
The Right-click Menu for a Part
in an Assembly File

MUST KNOW: By default, the first part that is placed into an assembly file will be grounded. A grounded part is a part that has all of it degrees of freedom removed and is fully constrained in that file. It is important to ground at least one part in every assembly since if no parts are grounded, the assembly can be moved around model space. Once one part is grounded, the other parts can be constrained to it making their movement relative to the grounded part.

USER TIP: The Browser bar contains a great deal of information about the active assembly file. The grounded part has a Push Pin icon. If a part is disabled, it will display a green part icon and if its visibility is disabled, the part icon will be grey as you can see in the figure.

WORKALONG: Creating an Assembly

Step 1

Check the default project and if necessary, set it to Inventor Course.

Step 2

Using the NEW command, enable the English tab and select the temple file: Modules Assembly (in).iam. Save and name the assembly: Inventor workalong 22-1. (Figure Step 2)

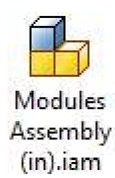


Figure Step 2

Step 3

Enter the PLACE COMPONENT command. In the Place Component dialogue box insert part files: Inventor Workalong 22-1A.ipt and Inventor Workalong 22-1B.ipt into the assembly file. (Figure Step 3A and 3B)

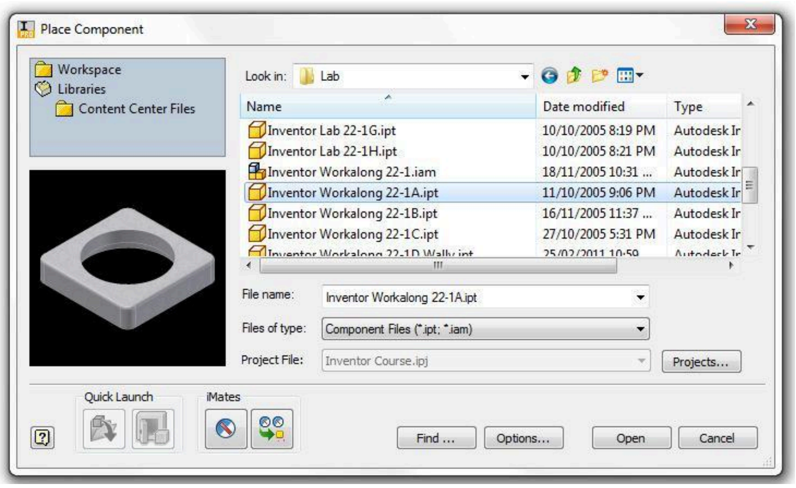


Figure Step 3A [Click to see image full size]

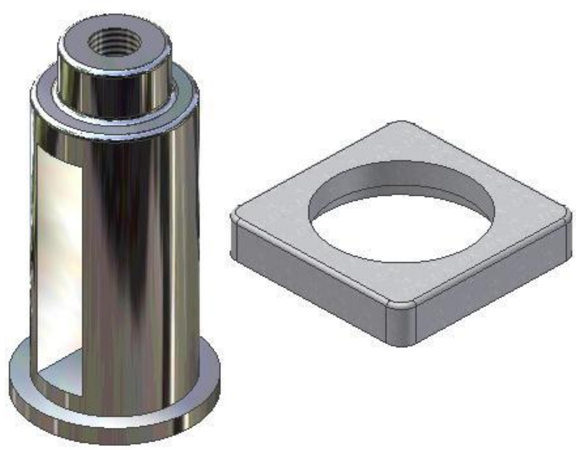


Figure Step 3B

USER TIP: By default, the PLACE COMPONENT command places multiple parts. To insert the part more than once into the assembly file, click once for each part. Ensure that you move the cursor so you are not inserting one part on top of another one. When you have inserted the required part(s), press Esc to exit the command.

AUTHOR'S COMMENTS: Where you locate the parts is not important but ensure that you insert each part only once.

Step 4

Disable the grounding of part 22-1A and enable the grounding of part 22-1B. (Figure Step 4A, 4B, and 4C)

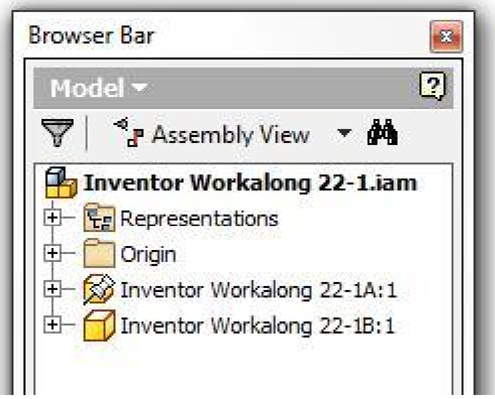


Figure Step 4A

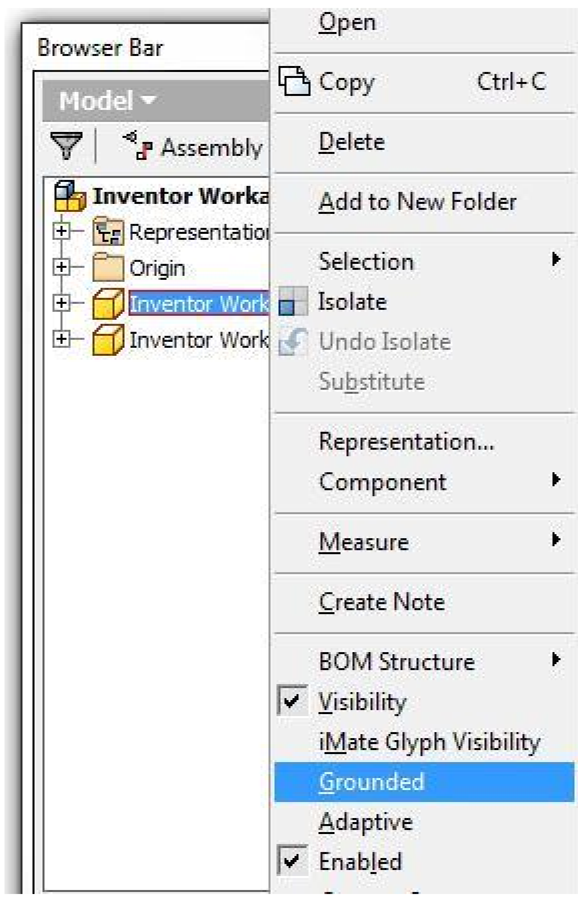


Figure Step 4B

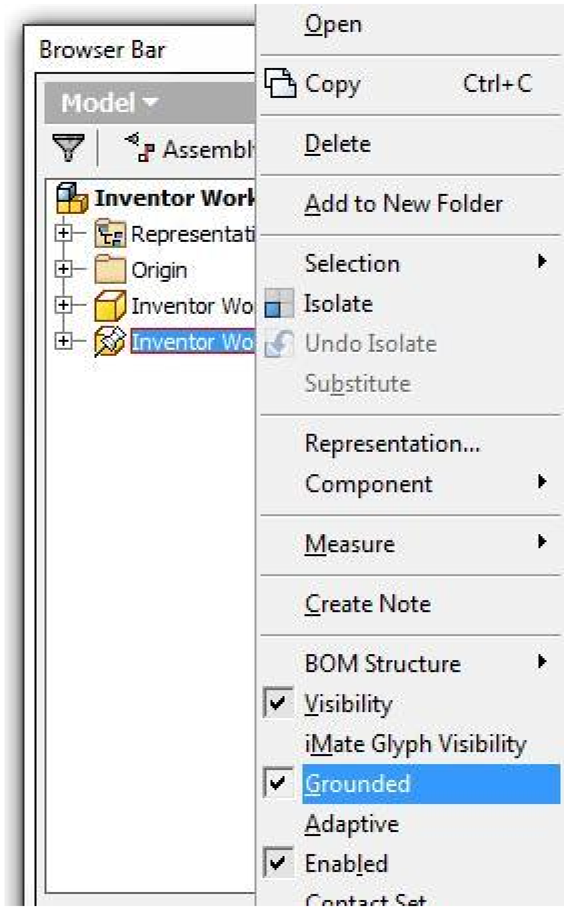


Figure Step 4C

Step 5

Enter the PLACE CONSTRAINT command. In the Place Constraint dialogue box set Type to Mate, Selections to 1, Offset to 0.000 and Solution to Mate. (Figure Step 5)

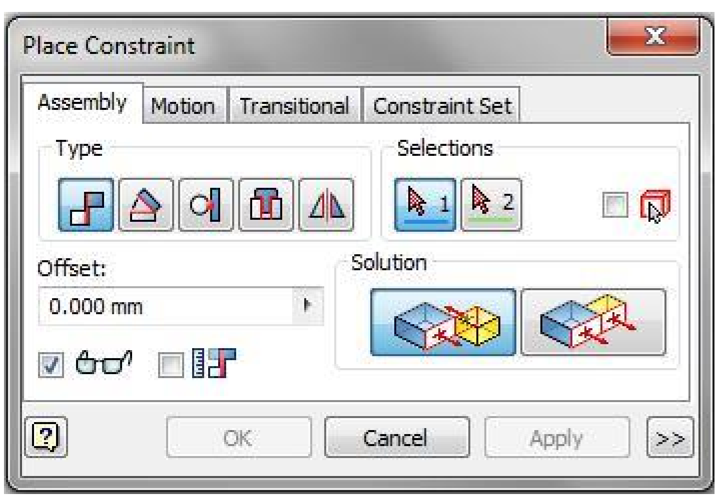


Figure Step 5

Step 6

Move the cursor onto the part 22-1A and move it until the Centerline constraint symbol appears. When it displays as shown in the figure, select it. Move the cursor onto the part 22-1B and do the same. Part 22-1A will move onto Part 22-1B. Click Apply. (Figure Step 6A, 6B, and 6C)

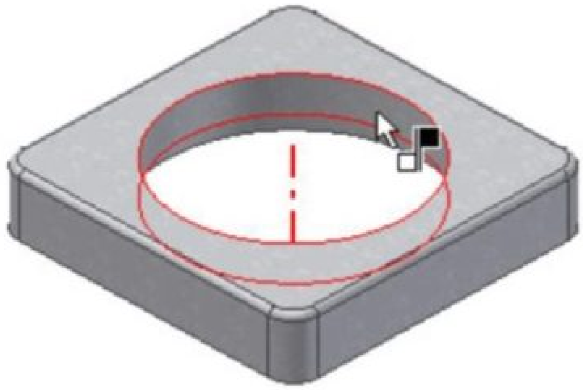


Figure Step 6A

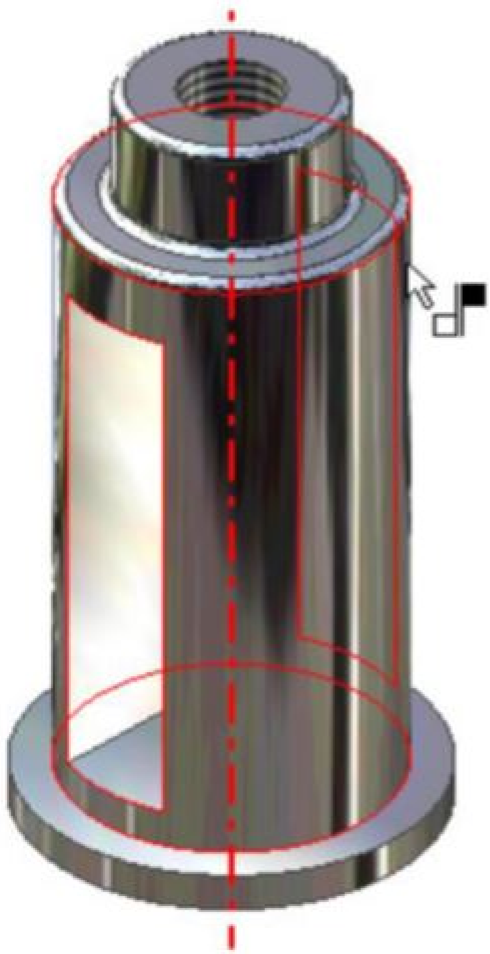


Figure Step 6B

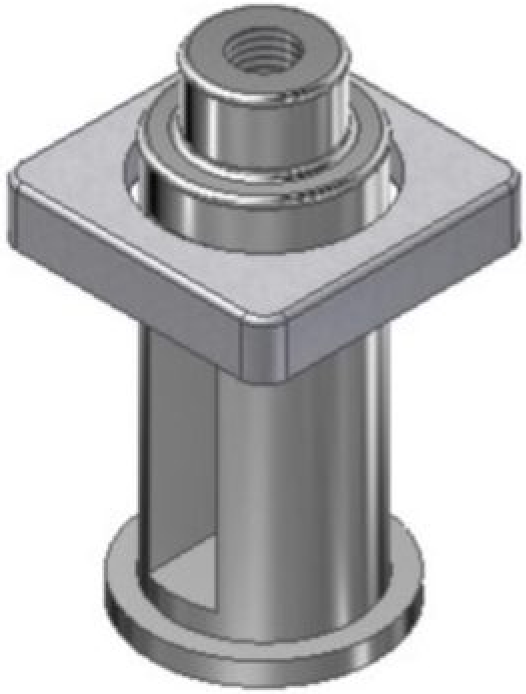


Figure Step 6C

> **AUTHOR'S COMMENTS:** Since part 22-1B is the grounded part, it will remain stationary and part 22-1A will move.

Step 7

Move the cursor onto the top of the base on part 22-1B as shown in the figure. The mate arrow will display pointing up. Press F4 and rotate the model. Select the mate constraint for the bottom of the part 22-1A. (Figure Step 7A, 7B, and 7C)

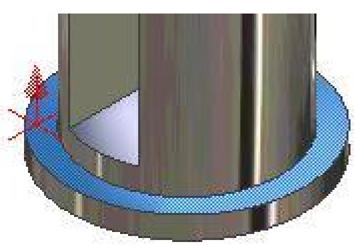


Figure Step 7A

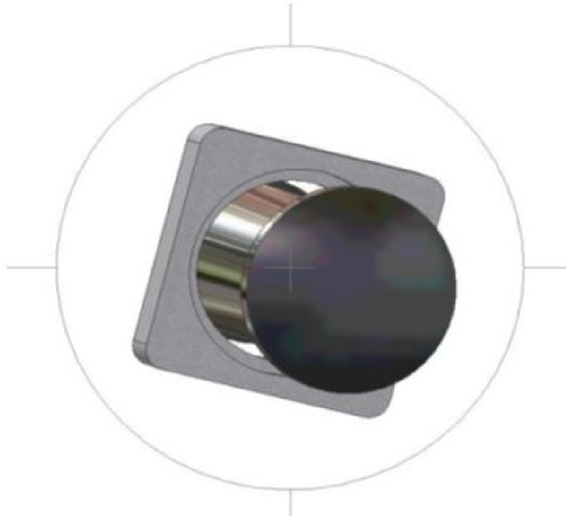


Figure Step 7B

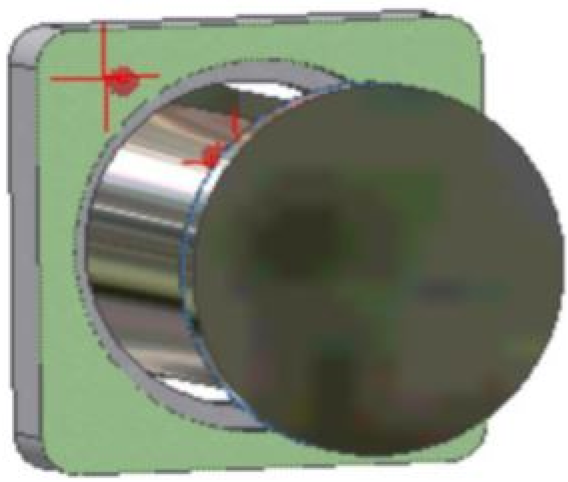


Figure Step 7C

Step 8

Press F6 to change the Home view. (Figure Step 8)

Figure Step 8

Step 9

Enter the PLACE COMPONENT (P) command and insert part 22-1C, 22-1D, 22-1E, and 22-1F. Ensure that you place each part only once. (Figure Step 9)

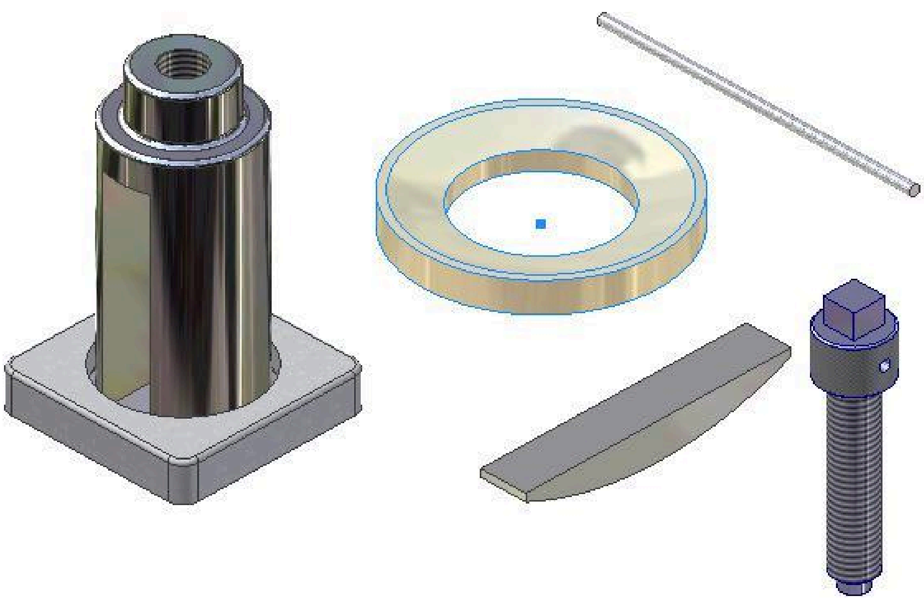


Figure Step 9

Step 10

Using what you just learned, assemble and constrain part 22-1C. (Figure Step 10)

Figure Step 10

Step 11

Using the PLACE CONSTRAINT command, constrain the arc in part 22-1D to the arc in part 22-1C. (Figure Step 11)

Figure Step 11

Step 12

In the Offset box, set the offset to 0.015. Enable the Pick Part First box. See the Author's Comments below. (Figure Step 12)

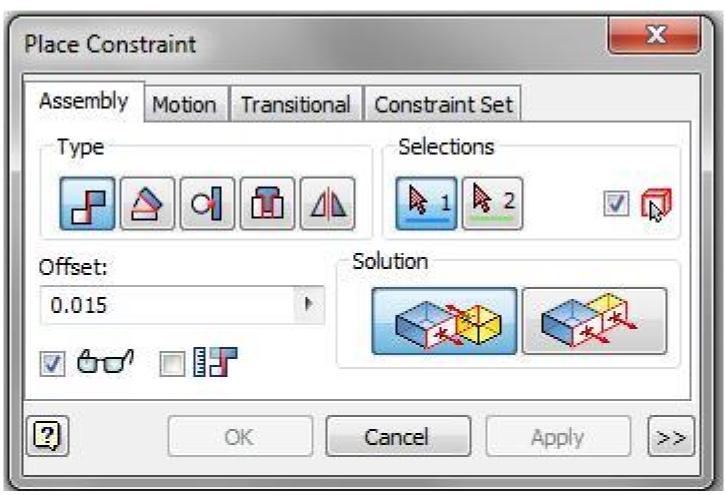


Figure Step 12

AUTHOR'S COMMENTS: The offset number was calculated by subtracting the width of the slot (0.75) and the width of the wedge (0.72) and then dividing by 2. That will centre the Wedge in the slot of the Post.

AUTHOR'S COMMENTS: When the Pick Part First box is enabled, the PLACE CONSTRAINT command allows the user to select the part first and then the constraint to be used. Sometimes it is easier to use the two pick method.

Step 13

To constrain the wedge in the slot, select one side of the slot and one side of the wedge as the mate surfaces. (Figure Step 13A, 13B, and 13C)

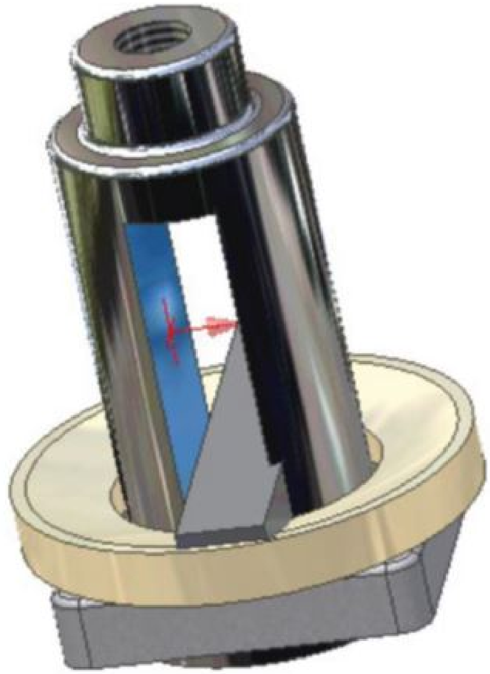


Figure Step 13A

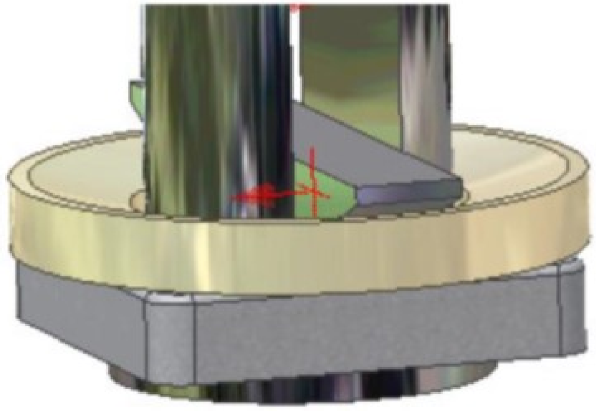


Figure Step 13B

Figure Step 13C

Step 14

Using what you just learned, constrain the last two parts to complete the assembly. (Figure Step 14)

Figure Step 14

AUTHOR'S COMMENTS: You will have to constrain both parts with a centerline constraint and an offset constraint.

Step 15

Save and close the file.

Key Principles

Key Principles in Module 22

1. An assembly file contains the information required to assemble two or more part files to create the assembled model. An assembly file has the file extension .iam.
2. An assembly file does do not contain any of the part files that are placed in the assembled model. It simply contains a reference to the part files. If an assembly file (.iam) is sent to a client or an associate, the part files (.ipt) that were placed in the assembly file must also be included otherwise they will not display when the assembly file is opened.
3. By default, the first part that is placed into an assembly file will be grounded. A grounded part is a part that has all of it degrees of freedom removed and is fully constrained in that file.
4. There are many different constraints used when creating an assembly. A mate constraint constrains two assembled parts to one another by mating their centerlines and/or by mating a plane on one part to a plane on the other part.

Lab Exercise 22-1

Time allowed: 180 minutes.

Part Name	**Project**	**Units**	**Template**	**Color**	**Material**
Inventor Lab 22-1	Inventor Course	Millimeters	See Below	See Below	See Below

Step 1

Create the following parts.

Step 2

Each part must have its own file. In each part, ensure that you do the following:
A Select your own location of X0Y0Z0.
B Draw the necessary sketches and extrude or revolve them to produce the solid model shown. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches. (Figure Step 2A and 2B)

C Apply the colour and material shown.

AUTHOR'S COMMENTS: Ensure that you draw each part in the correct orientation so that they can be assembled together.

Part: Tire

Part Name: Inventor Lab 22-1A
Template: Metric-Modules Part (mm).ipt
Color: Rubber – Green – Version 1.1
Material: Rubber

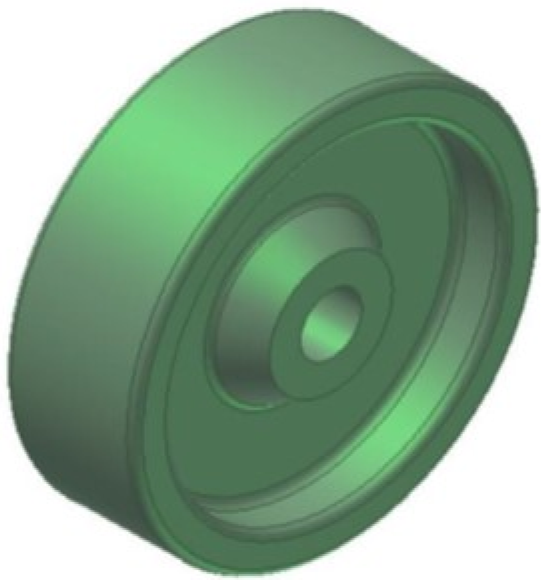


Figure Step 2A
Solid Model – Home View

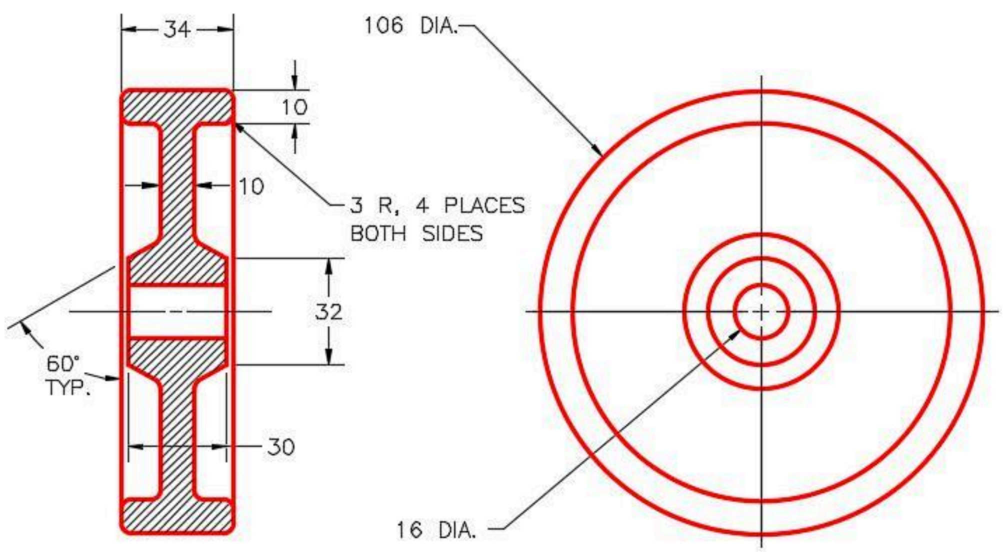


Figure Step 2B
Dimensioned Multiview Drawing

Part: Frame
Part Name: Inventor Lab 22-1B
Template: Metric-Modules Part (mm).ipt
Color: Aluminum – Polished
Material: Steel (Figure Step 2C and 2D)

Figure Step 2C
Solid Model – Home View

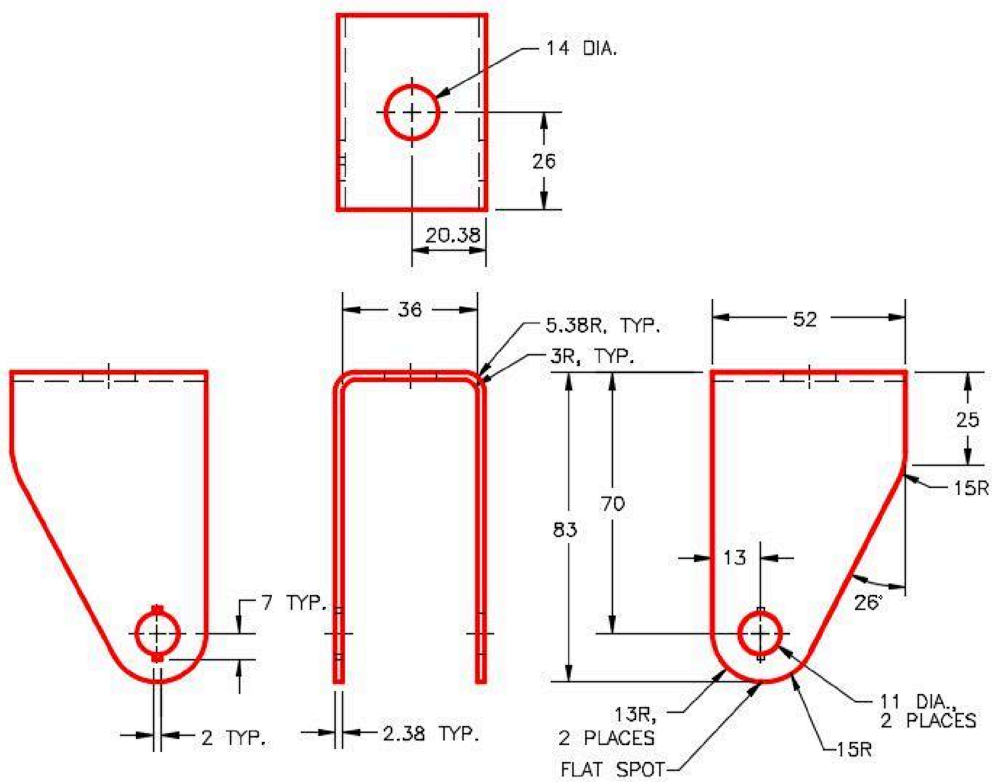


Figure Step 2D
Dimensioned Multiview drawing [Click to see image full size]

Part: Bushing
Part Name: Inventor Lab 22-1C
Template: Metric-Modules Part (mm).ipt
Color: Brass – Satin
Material: Brass (Figure Step 2E and 2F)

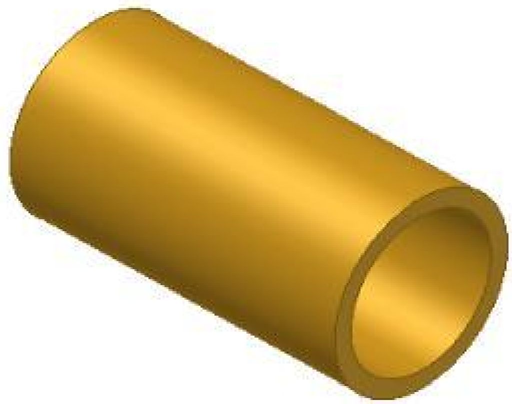


Figure Step 2E
Solid Model –
Home View

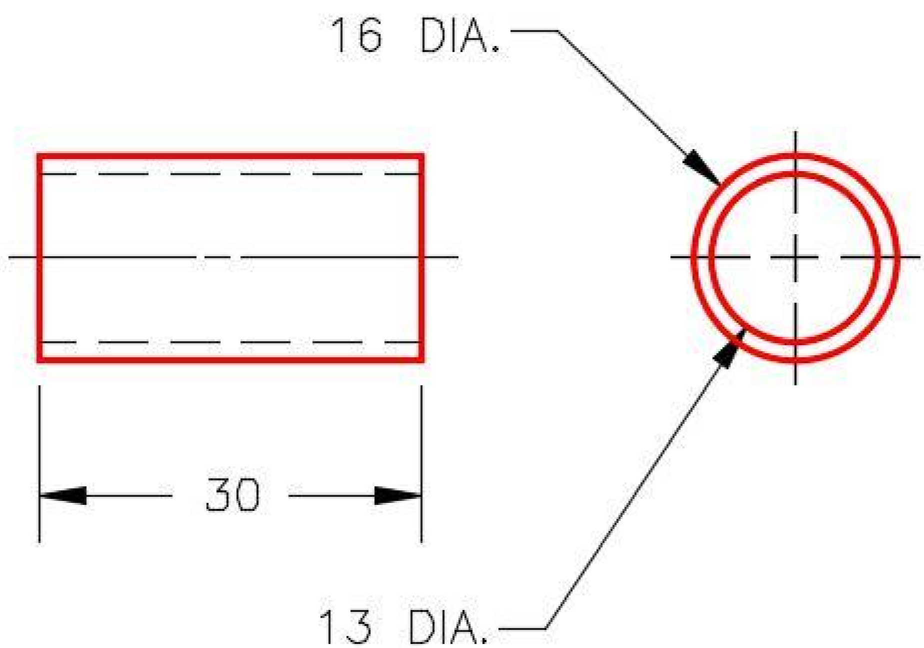


Figure Step 2F
Dimensioned Multiview Drawing

Part: Pin
Part Name: Inventor Lab 22-1D
Template: Metric-Modules Part (mm).ipt
Color: Semi – Polished
Material: Steel (Figure Step 2G, 2H and 2J)

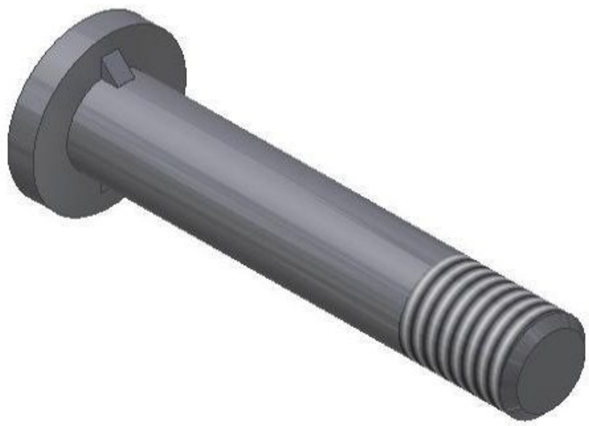


Figure Step 2G
Solid Model – Home View

Figure Step 2H
Solid Model – Orbited View

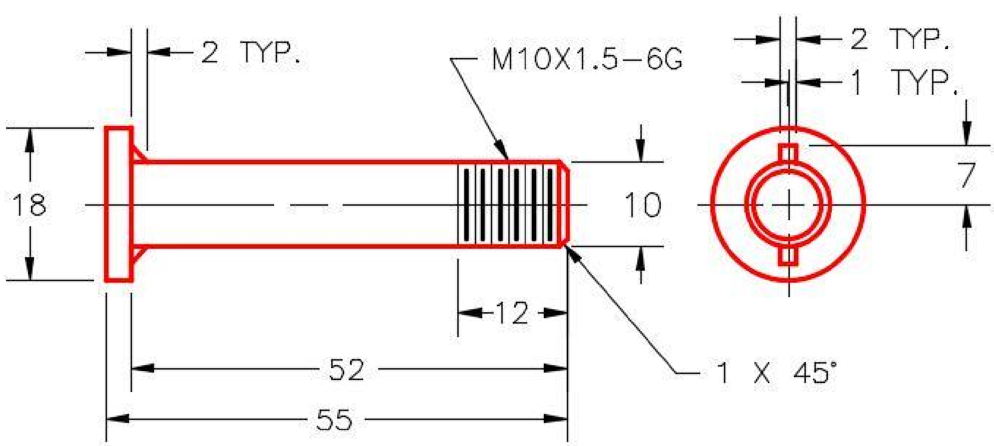


Figure Step 2J
Figure Step 2H
Solid Model – Orbited View
Dimensioned Multiview Drawing [Click to see image full size]

Part: 10 mm Nut
Part Name: Inventor Lab 22-1E
Template: Metric-Modules Part (mm).ipt
Color: Semi – Polished
Material: Steel (Figure Step 2K and 2L)

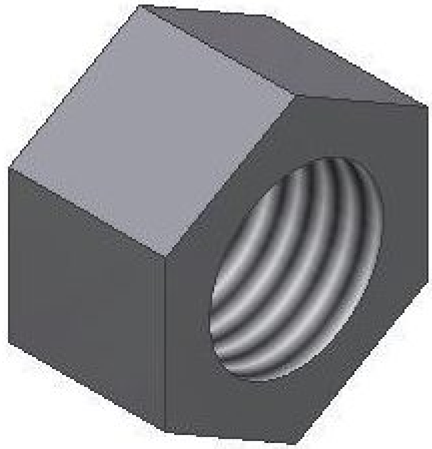


Figure Step 2K
Solid Model – Home View

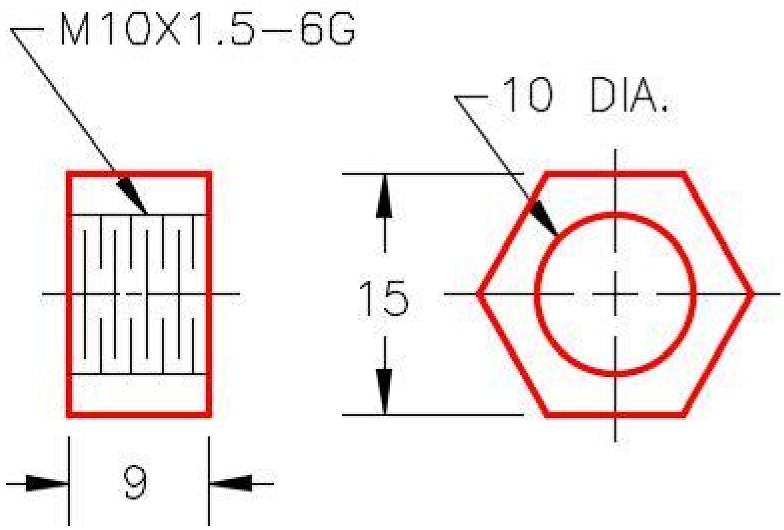


Figure Step 2L
Dimensioned Multiview Drawing

Part: Bolt
Part Name: Inventor Lab 22-1F
Template: Metric-Modules Part (mm).ipt
Color: Semi – Polished
Material: Steel (Figure Step 2M and 2N)

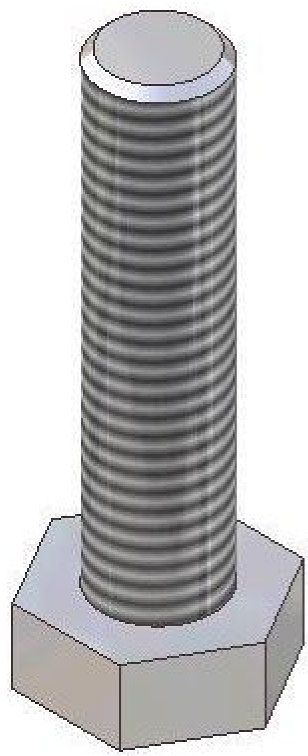


Figure Step 2M Solid Model – Home View

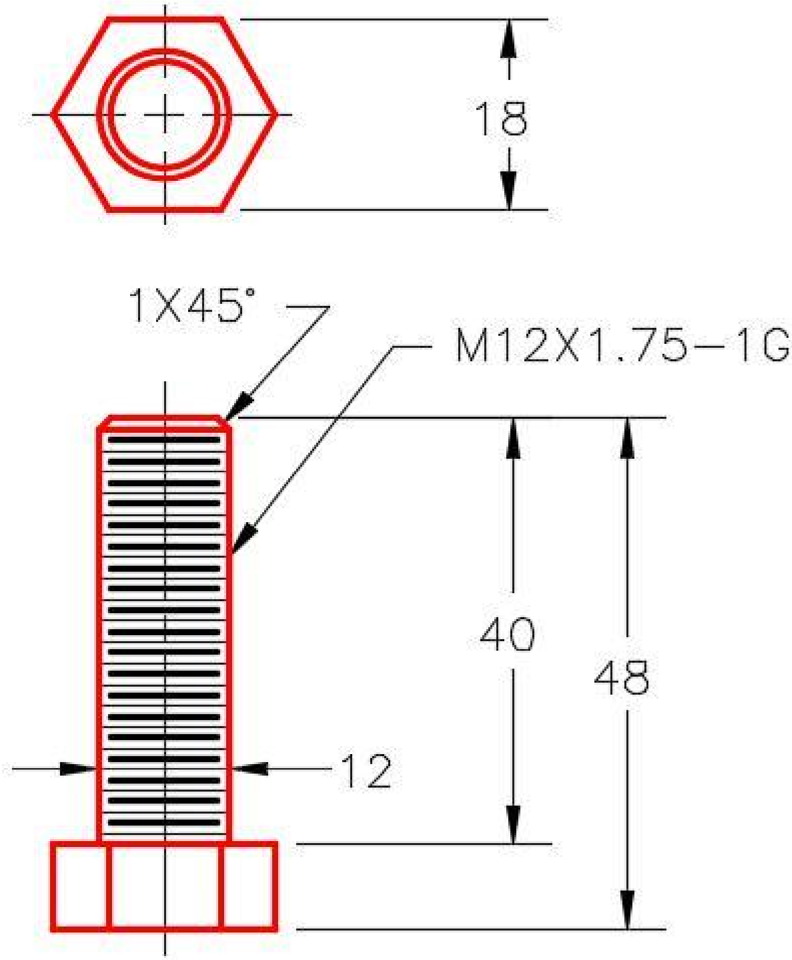


Figure Step 2N
Dimensioned Multiview Drawing

Part: 12 mm Nut
Part Name: Inventor Lab 22-1G
Template: Metric-Modules Part (mm).ipt
Color: Semi-Polished
Material: Steel (Figure Step 2P and 2Q)

Figure Step 2P
Solid Model – Home View

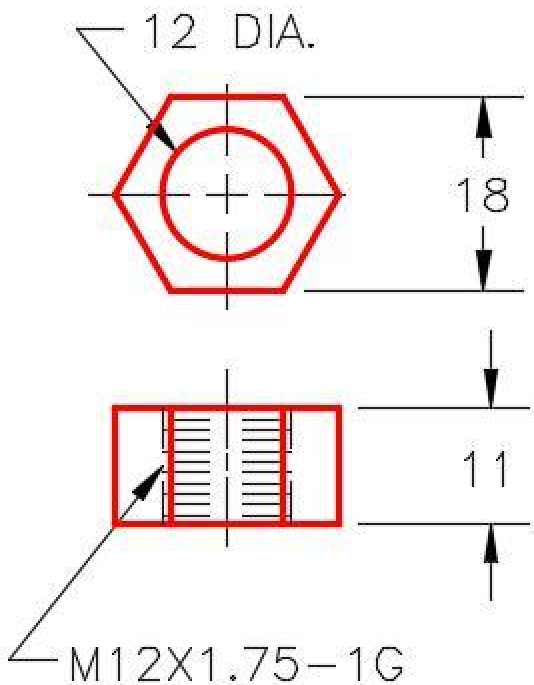


Figure Step 2Q
Dimensioned Multiview Drawing

Part: Washer
Part Name: Inventor Lab 22-1H
Template: Metric-Modules Part (mm).ipt
Color: Semi-Polished
Material: Steel (Figure Step 2R and 2S)

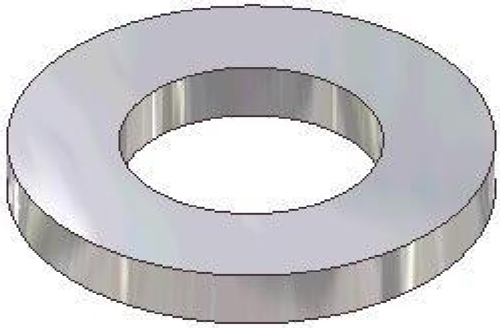


Figure Step 2R
Solid Model –
Home View

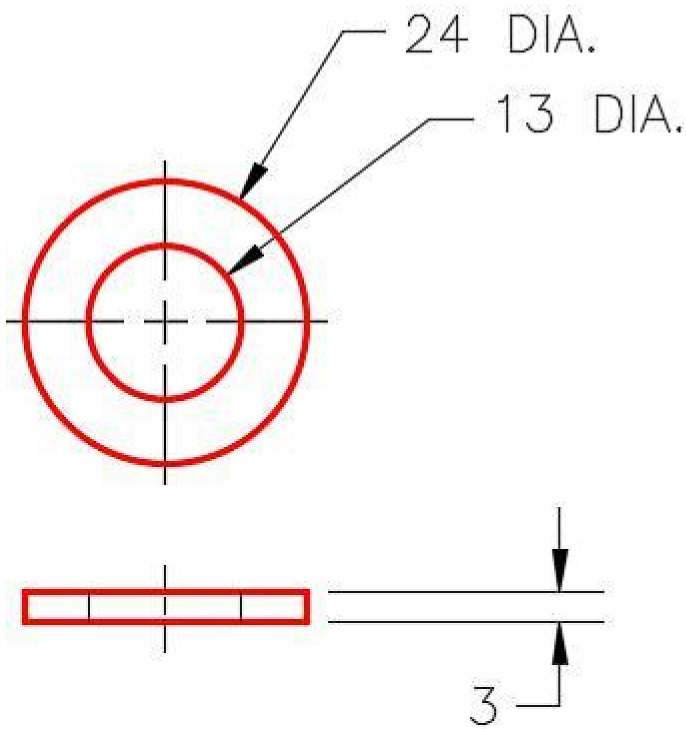


Figure Step 2S
Dimensioned Multiview Drawing

Step 3

Open a new assembly file.

Assembly: Caster
File Name: Inventor Lab 22-1.iam
Template: Metric – Modules Assembly (mm).iam

Step 4

Set the Tire as the grounded part.

Step 5

Start with assembling the Bushing into the Tire first. (Figure Step 5)

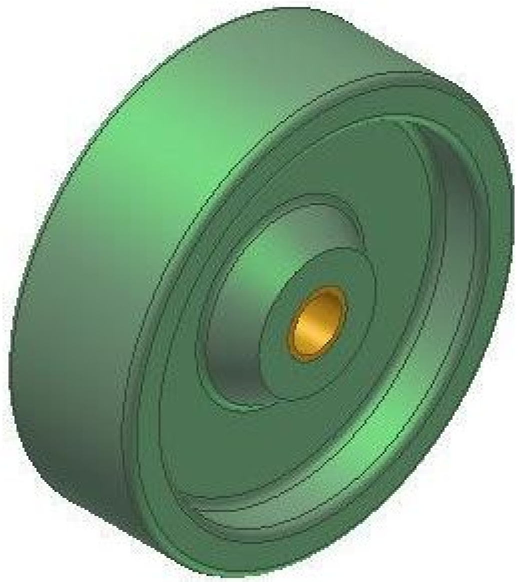


Figure Step 5

Step 6

Assemble the remaining parts to complete the assemble file. (Figure Step 6A and 6B)

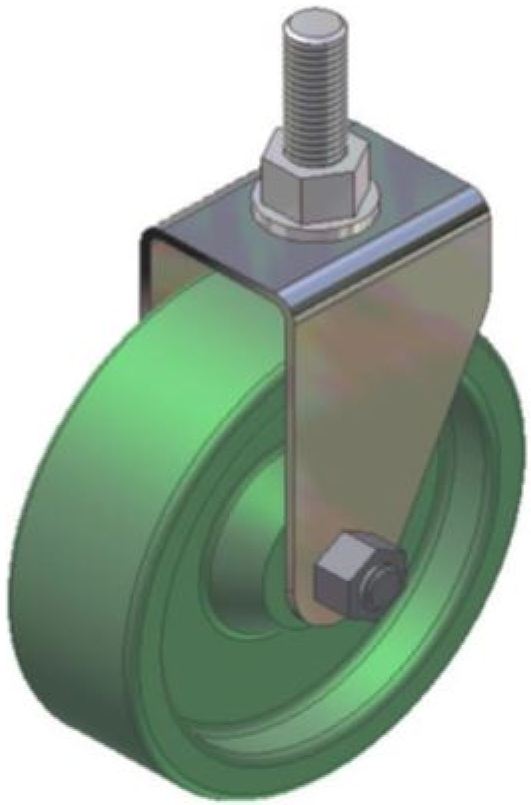


Figure Step 6A Assembled Model – Home View

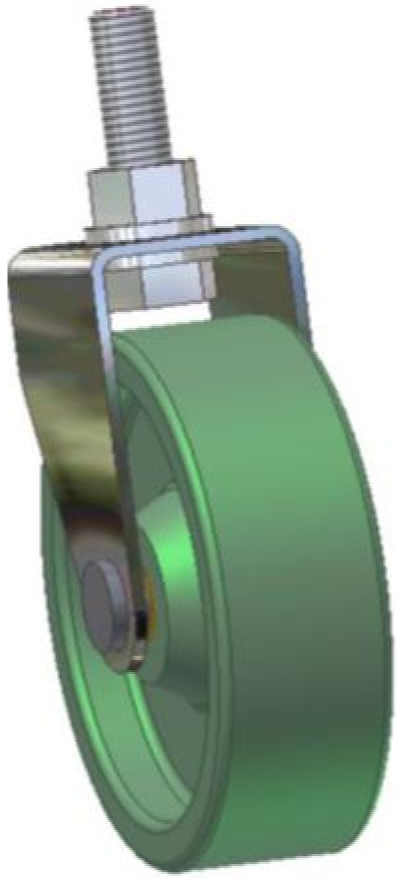


Figure Step 6B Assembled Model – Orbited View

AUTHOR'S COMMENTS: There are 2 washers. One above and one below the frame.

Module 23 Presentation Files

Learning Outcomes

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

1. Describe a presentation file, an exploded assembly, and an animation.
2. Describe and apply the CREATE VIEW, PRECISE VIEW ROTATION, TWEAK COMPONENT, and ANIMATE commands to create, tweak, and play the animation of an exploded assembly.

Presentation Files

After the model is assembled in the assembly file, a presentation file can be created using it. A *presentation file* is a file in which a view of an assembly is exploded and can be animated. It can also be rotated so that is can be viewed from different angles. A presentation file is created using the NEW command and selecting a presentation template. A presentation file has the extension .ipn. IPN is an acronym for Inventor PreseNtation.

Exploded Views

An *exploded view* shows the assembly as if it were dismantled and the components of the assembly shown in the order and orientation that they fit together to create the assembly. Assemblies can be exploded automatically by Inventor, tweaked manually, or a combination of the two methods, complete with trails. A *tweak* is the distance that the part is moved from the grounded part while the *trails* are the lines in an exploded view that show the relationship of the component to the assembly. Combined, they indicate the direction and distance that a component was moved to create the view. Exploded assembly drawings are used in design presentations, catalogues, and assembly instruction. Figure 23-1 shows an exploded and rotated view of an assembly.

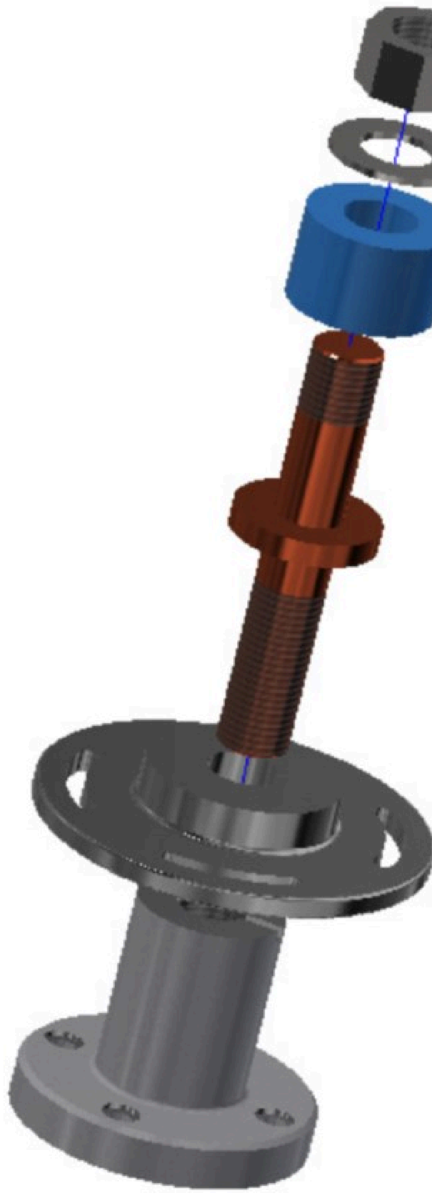


Figure 23-1
An Exploded and Rotated
Presentation Model

Animations

An *animation* is a series of images of an exploded assembly showing each components tweak at a set interval speed making a small motion picture of an assembly or a disassembly.

Tweaking

Tweaking is the process of moving the components from the grounded component. The tweak is the distance measured from the grounded unit. In the automated explosion method, the tweak distance is same for each part. If the tweak distance is set to 35, that means the first part is 35 mm from its location and the second part is 70 mm etc. This can be seen if the children in the Browser bar are expanded as shown in Figure 23-2. To tweak components manually, use the TWEAK COMPONENT command.

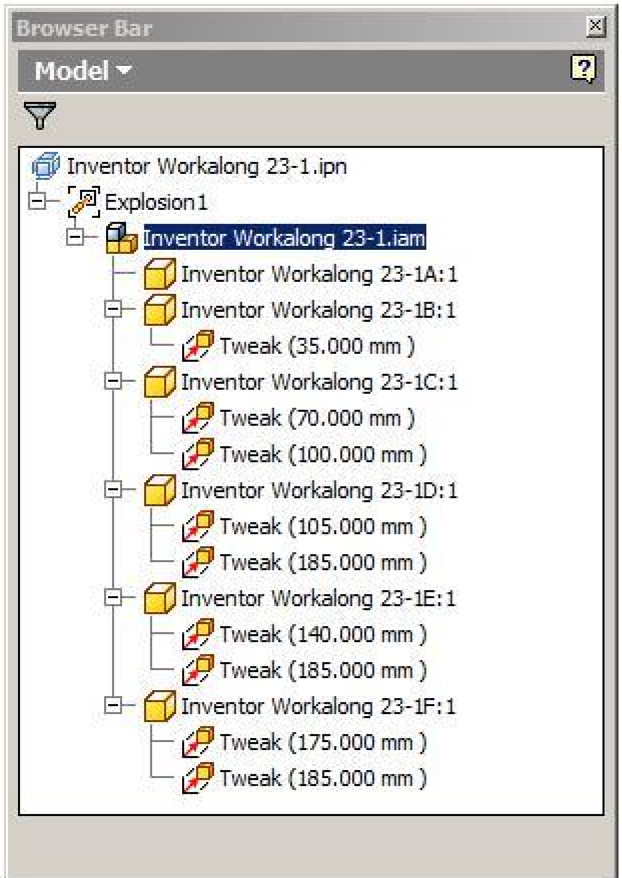


Figure 23-2
Tweaking

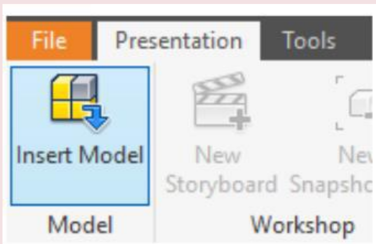
Animating the Exploded Assembly

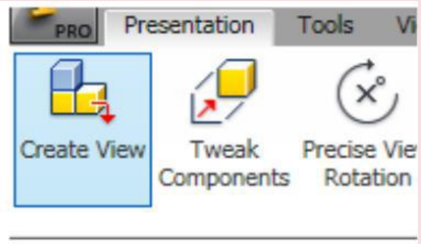
After the exploded assembly is tweaked, it can animated to show the assembly and the disassembly of the model. An animation is simply a series of *frames* or pictures of the assembly displayed one frame at a time. The amount of time between frames is called the *interval*. The larger the interval, the slower the animation and the shorter the interval, the faster the animation. The number of repetitions can also be set.

Inventor Command: CREATE VIEW or INSERT MODEL

The CREATE VIEW or INSERT MODEL commands are used to insert the model of the assembly file.

Shortcut: **none**

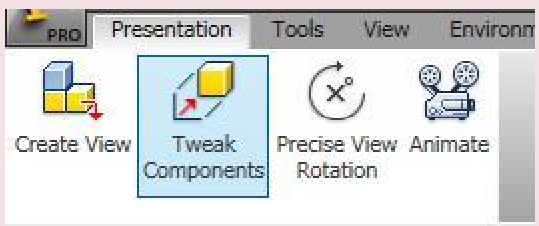




Inventor Command: TWEAK COMPONENTS

The TWEAK COMPONENTS command is used to move a component farther or closer from the grounded component to create or edit an exploded view of the assembly. It can also be used to rotate the components.

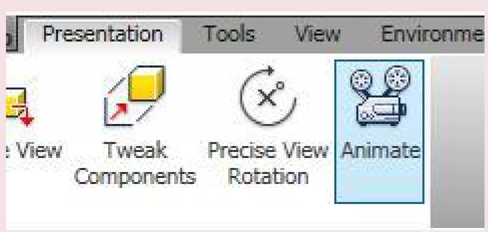
Shortcut: **T**



Inventor Command: ANIMATE

The ANIMATE command is used to set the parameters and play an animation of an assembly or disassembly of a assembled model.

Shortcut: **none**



MUST KNOW: A presentation file is a file in which a view of an assembly is created, rotated, exploded, and animated. To create a presentation file, use the NEW command and select a presentation template. A presentation file has the extension .ipn. IPN is an acronym for Inventor Presentation.

WORK ALONG: Creating an Assembly and Presentation File

Step 1

Check the default project and if necessary, set it to Inventor Course.

Step 2

Use the following instructions to complete all parts in this workalong. Create the following parts and ensure that you do the following:
A Each part must be saved in its own .ipt file.
B Project the Center Point onto the Base sketch plane and note the location of X0Y0Z0.
C Draw the necessary sketches and extrude or revolve them to produce the solid model. Apply all of the necessary geometrical and dimensional constraints to fully constrain it.
D Apply the colour and material shown.

Step 3

Create the part file as follows:
Part: Base
Part Name: Inventor Workalong 23-1A Template: Metric-Modules Part (mm).ipt Color: Aluminum Cast Material: Aluminum 6061 (Figure Step 3A and 3B)

Template: Metric-Modules Part (mm).ipt

Color: Aluminum

Cast Material: Aluminum 6061 (Figure Step 3A and 3B)

Figure Step 3A
Solid Model – Home View

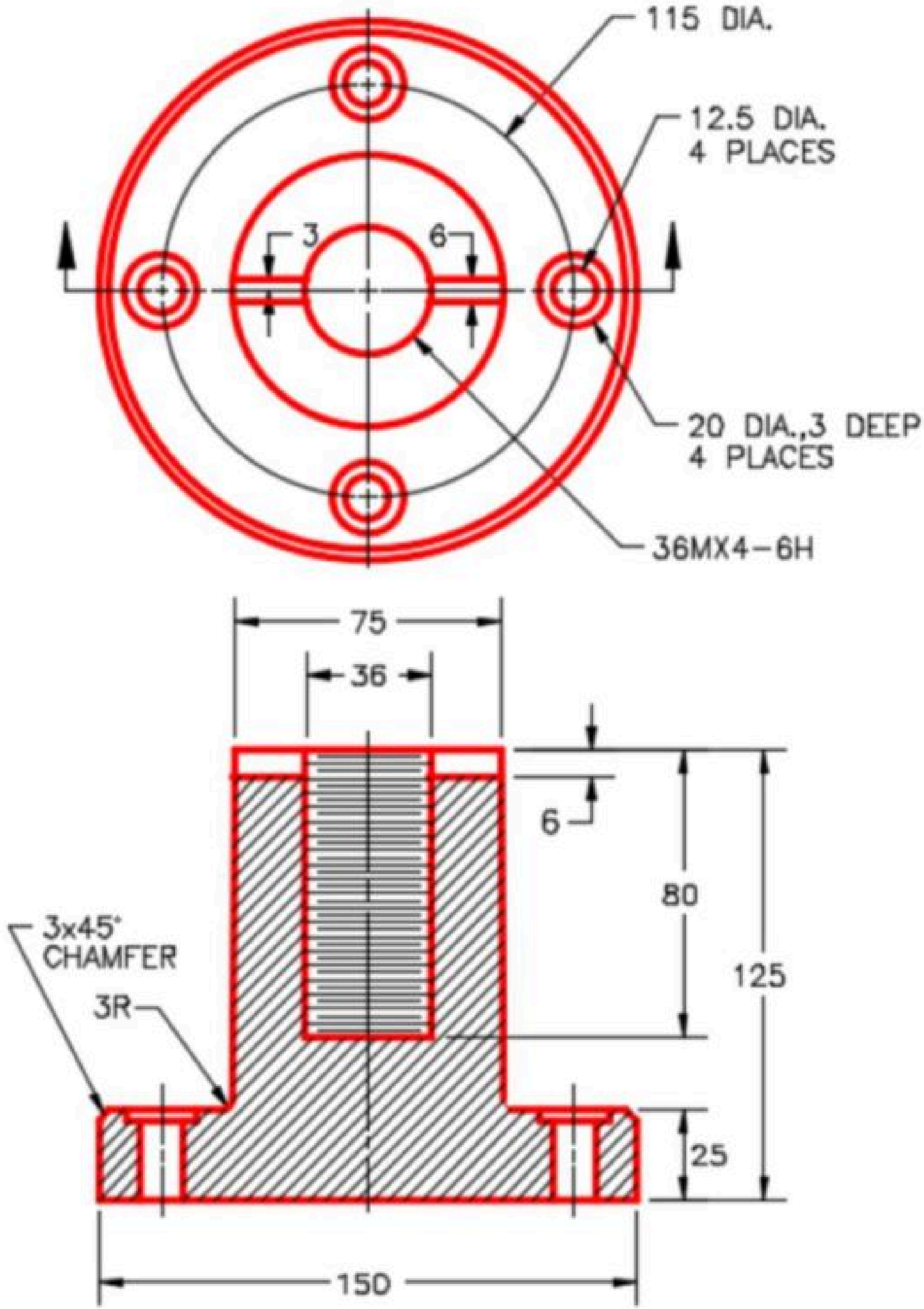


Figure Step 3B
Dimensioned Multiview Drawing [Click to see image full size]

Step 4

Change the colour of the machined faces to: Mirror.

Part: Slotted Wheel
Part Name: Inventor Workalong 23-1B
Template: Metric-Modules Part (mm).ipt
Color: Mirror
Material: Chrome – Polished Blue (Figure Step 4A, 4B, and 4C)

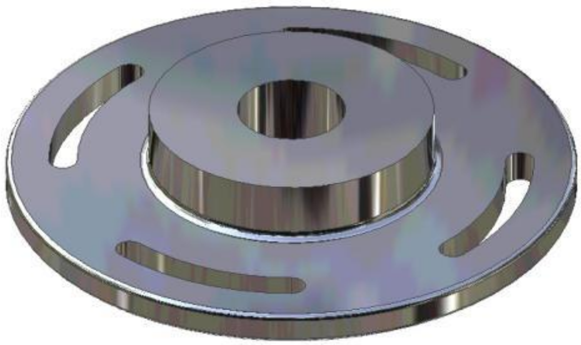


Figure Step 4A
Solid Model – Home View

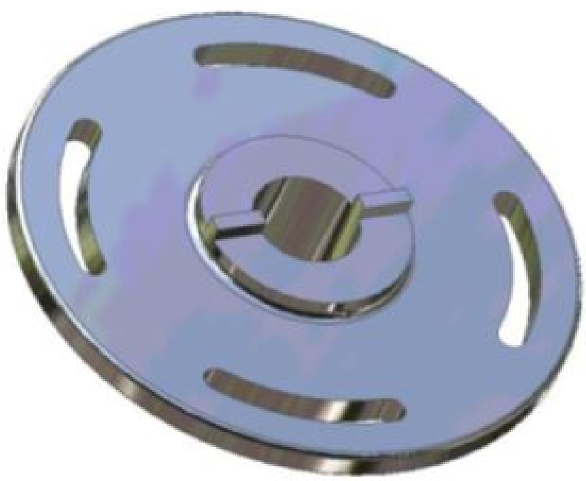


Figure Step 4B
Solid Model – Orbited View

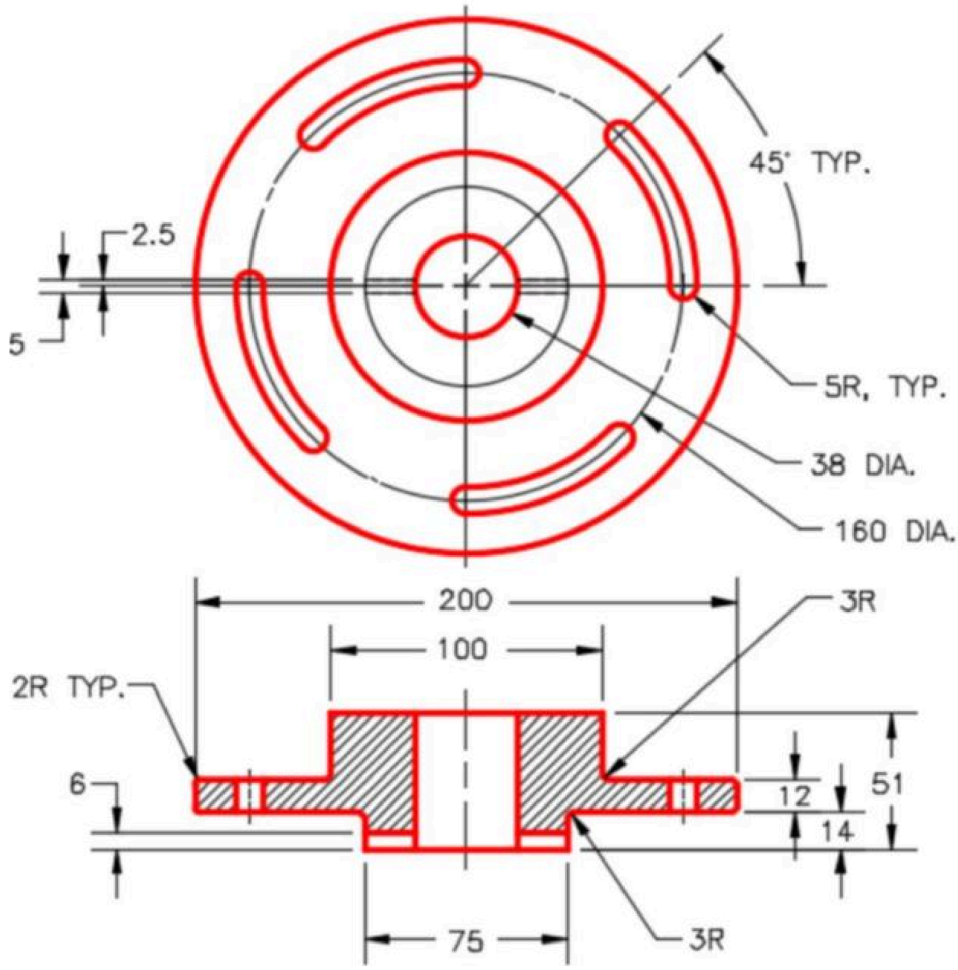


Figure Step 4C
Dimensioned Multiview Drawing

Part: Connecting Shaft
Part Name: Inventor Workalong 23-1C
Template: Metric-Modules Part (mm).ipt
Color: Copper – Polished
Material: Copper (Figure Step 4D and 4E)

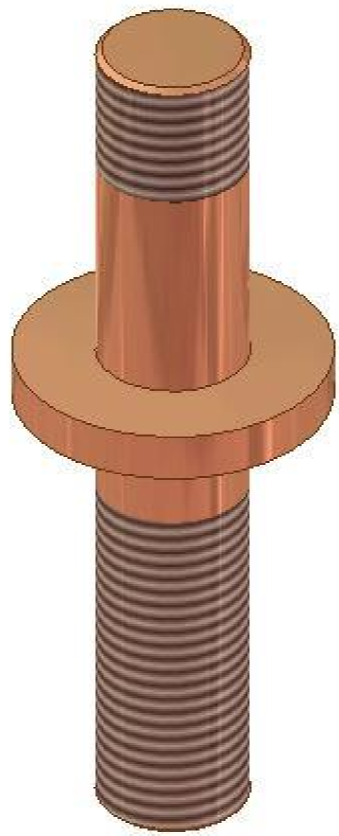


Figure Step 4D
Solid Model –
Home View

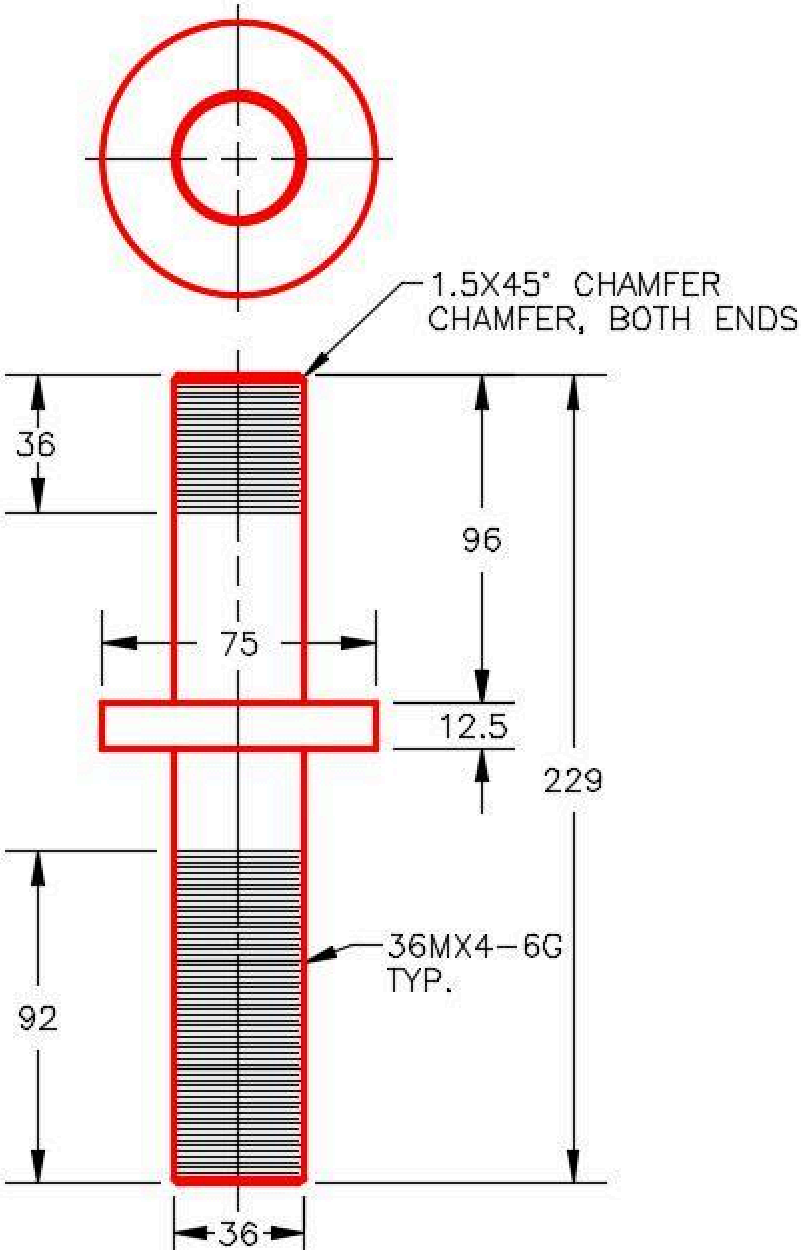


Figure Step 4E
Dimensioned Multiview Drawing

Part: Spacer
Part Name: Inventor Workalong 23-1D
Template: Metric-Modules Part (mm).ipt
Color: Blue – Wall Paint – Glossy
Material: ABS Plastic (Figure Step 4F and 4G)

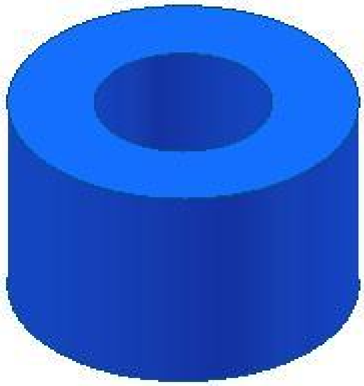


Figure Step 4F
Solid Model –
Home View

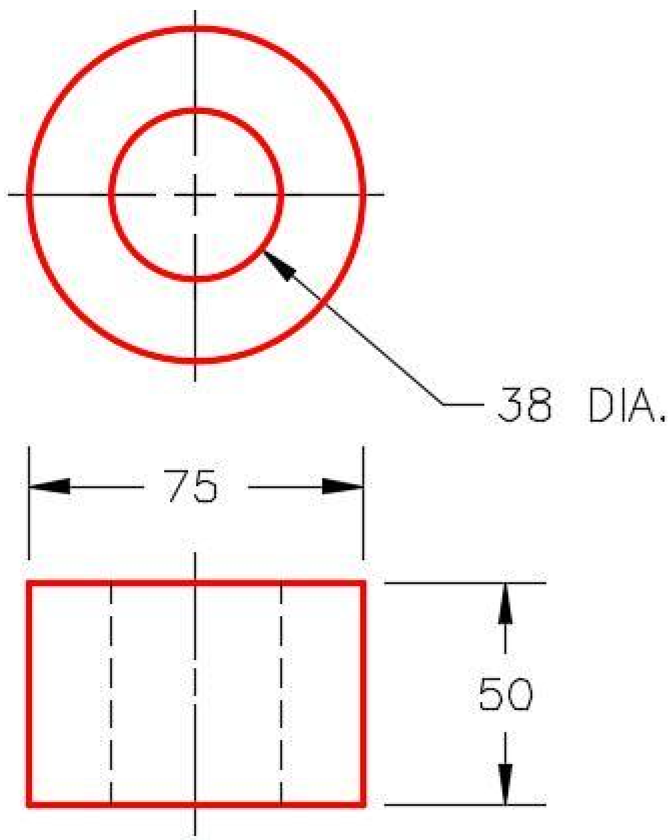


Figure Step 4G
Dimensioned
Multiview Drawing

Part: Washer
Part Name: Inventor Workalong 23-1E
Template: Metric-Modules Part (mm).ipt
Color: Steel – Polished
Material: Steel (Figure Step 4H and 4J)

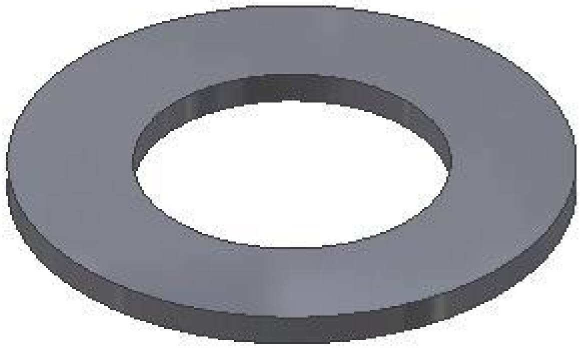


Figure Step 4H Solid Model – Home View

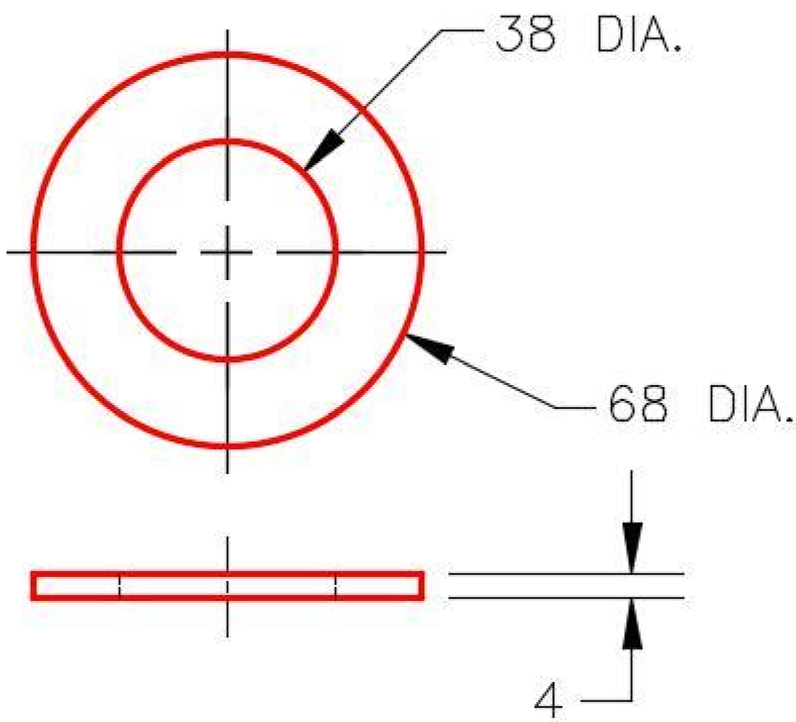


Figure Step 4J Dimensioned Multiview Drawing

Part: Nut
Part Name: Inventor Workalong 23-1F
Template: Metric-Modules Part (mm).ipt
Color: Steel – Polished
Material: Steel (Figure Step 4K and 4L)

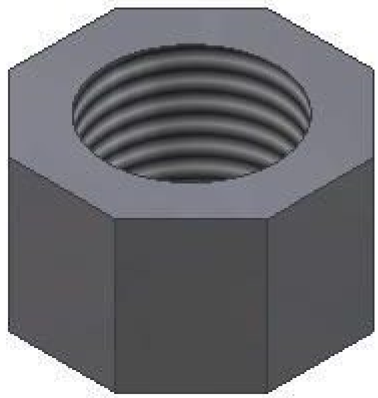


Figure Step 4K
Solid Model –
Home View

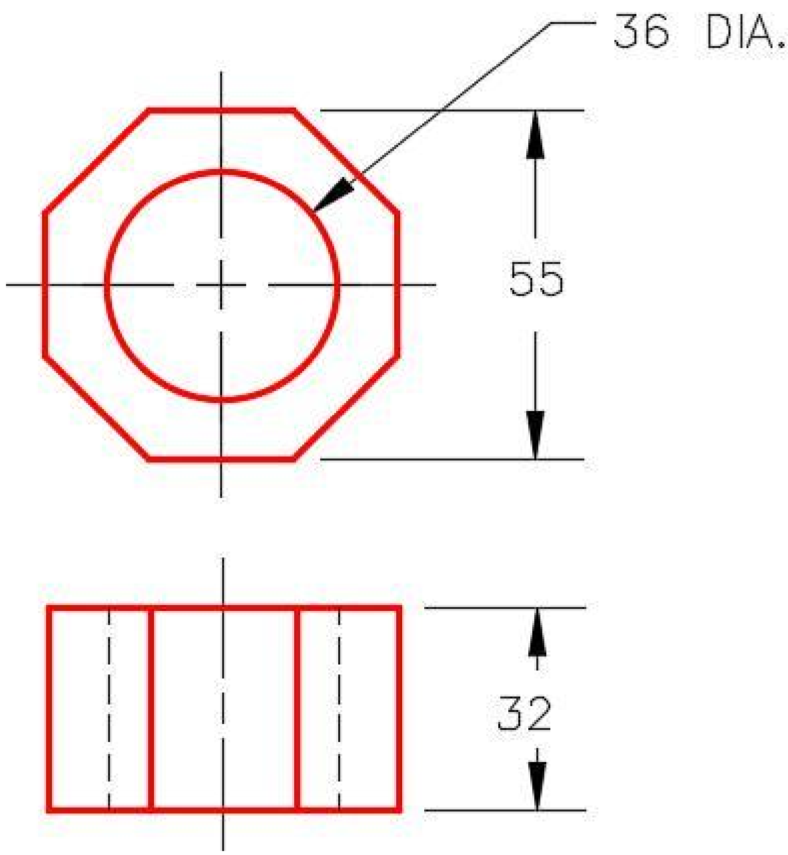


Figure Step 4L
Dimensioned
Multiview Drawing

Step 5

Assemble all of the parts that you just created as shown in the figures.
Assembly: Slotted Connector
Assembly Name: Inventor Workalong 23-1.iam
Template: Metric-Modules Assembly (mm).iam (Figure Step 5A and 5B)

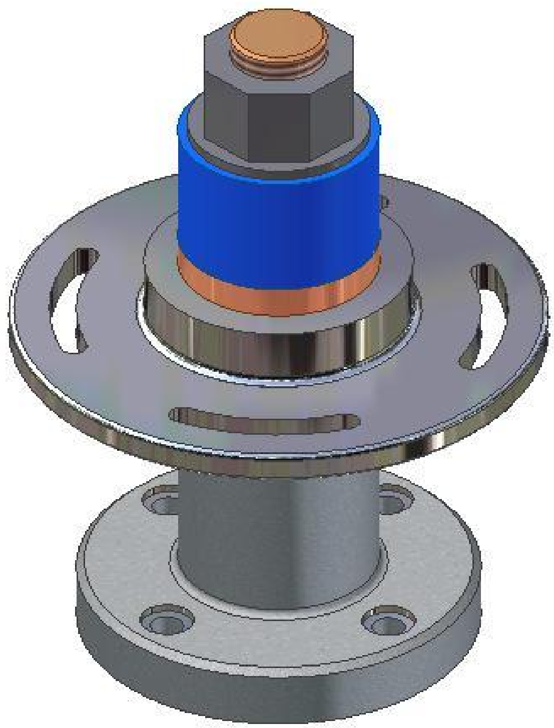


Figure Step 5A
Assembly – Home View

Figure Step 5B
Assembly – Orbited View

Step 6

Enter the NEW command and enable the Metric tab. Select the template file: Modules Presentation (mm).ipn (Figure Step 6)

Figure
Step 6

Step 7

Enter the CREATE VIEW command. It will open the Select Assembly dialogue box. (Figure Step 7)

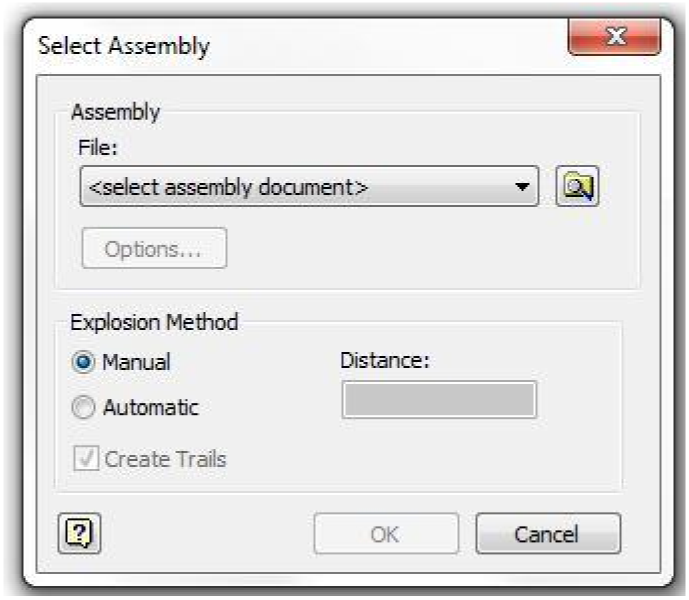


Figure Step 7

Step 8

Click the Search Folder icon at the end of the file name. It will open the Open dialogue box. In the Lab Exercises folder, select the file: Inventor Workalong 23-1.iam and click Open. (Figure Step 8)

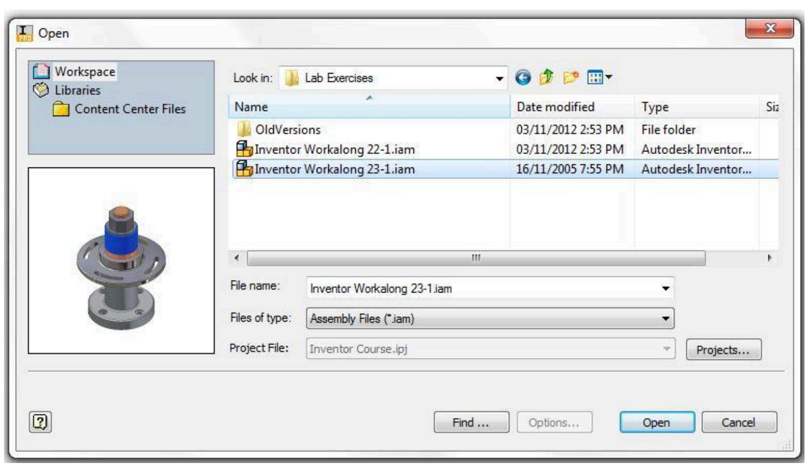


Figure Step 8 [Click to see image full size]

Step 9

In the Select Assembly dialogue box enable Automatic and Create Trails in the Explosion Method area. Enter the Distance of 35 mm. Click OK. The exploded assembly should appear as shown in the figure. (Figure Step 9A and 9B)

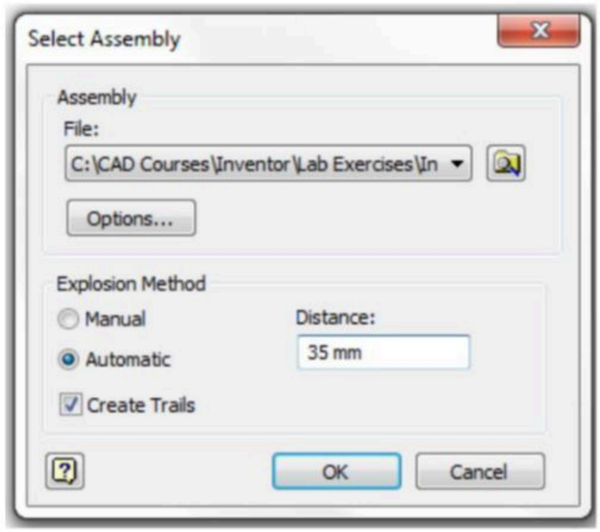


Figure Step 9A [Click to see image full size]

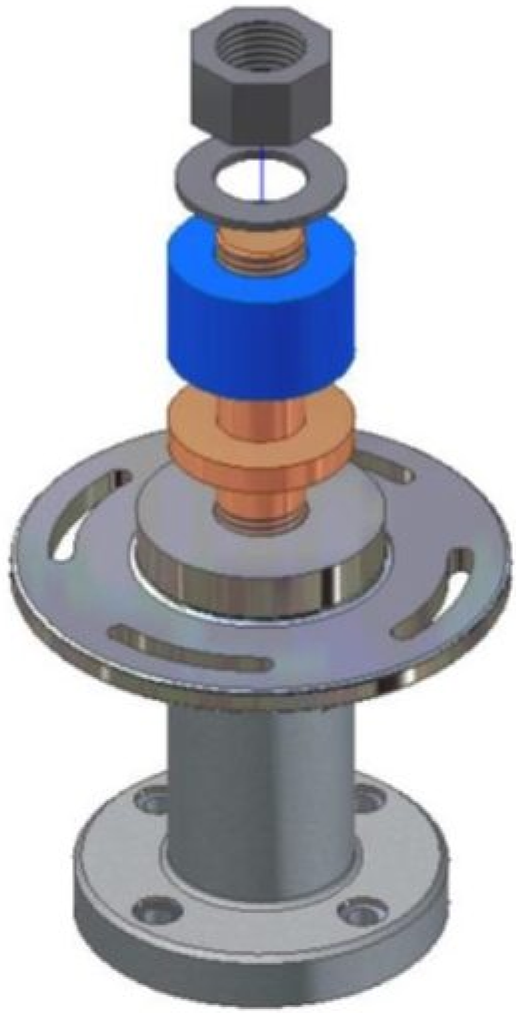


Figure Step 9B

Step 10

Enter the PRECISE VIEW ROTATION command. In the Incremental View Rotate dialogue box, set the Increment to 10. (Figure Step 10)

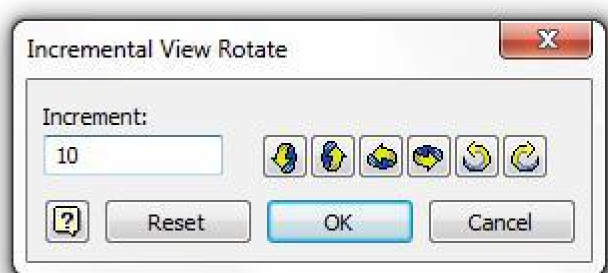


Figure Step 10

AUTHOR'S COMMENTS: The Increment setting sets the number of degrees the assembly will rotate per click.

Step 11

Click the Right Rotate icon twice. This will rotate the model 20 degrees to the right. Click the Up Rotate icon once. This will rotate the model 10 degrees upwards. (Figure Step 11)

Figure Step 11

Step 12

Enter the TWEAK COMPONENTS command. It will open the Tweak Component dialogue box. Set it as shown in the figure. (Figure Step 12)

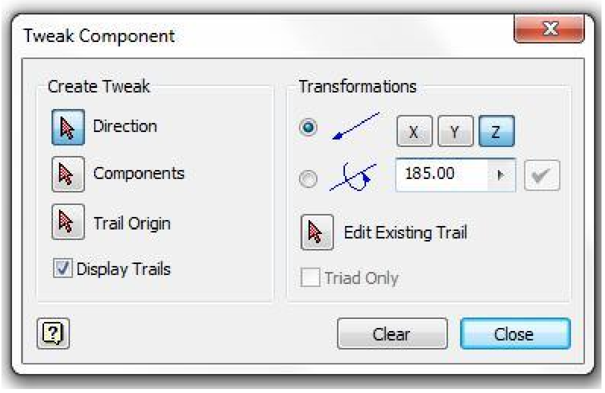


Figure Step 12

Step 13

Ensuring that Direction is enabled, zoom in and move the cursor on the top of the nut. When the direction icon is displayed as shown in the figure, click the left mouse button. (Figure Step 13A and 13B)

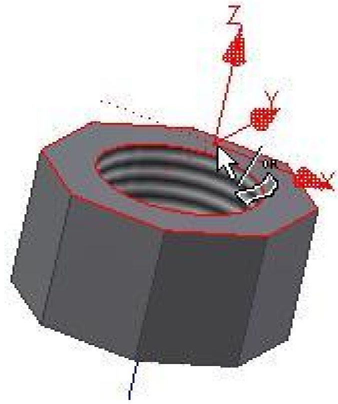


Figure Step 13A

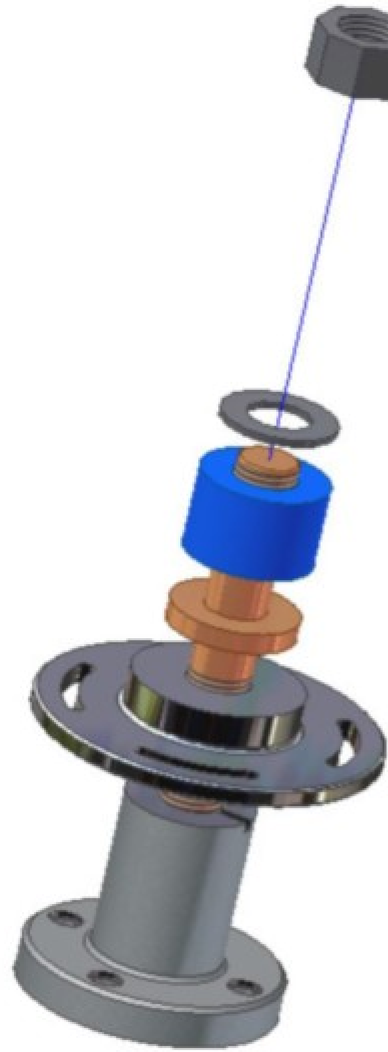


Figure Step 13B

AUTHOR'S COMMENTS: Since the Transformations is set to the Z direction and a distance of 185 mm the Nut will move 185 mm in the Z direction.

Step 14

Look at the Browser bar for the tweak you just created. It will show two tweaks for part:

Inventor Workalong 23-1F:1, which is the nut. The 175.000 mm tweak was created in the automatic explosion in Step 9 and the 185.000 mm from the manual tweak you just completed. (Figure Step 14)

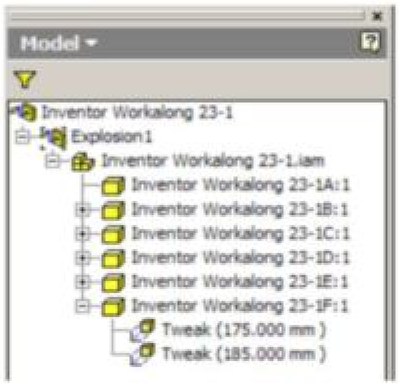


Figure Step 14

AUTHOR'S COMMENTS: The 175.00 mm is 5 X 35.00. The nut is the fifth part to move from the grounded part.

Step 15

Click the Clear button to clear the current settings. Using the Browser bar in Figure Step 15A, manually tweak the assembly to match the figure shown in Figure Step 15B. (Figure Step 15A and 15B)

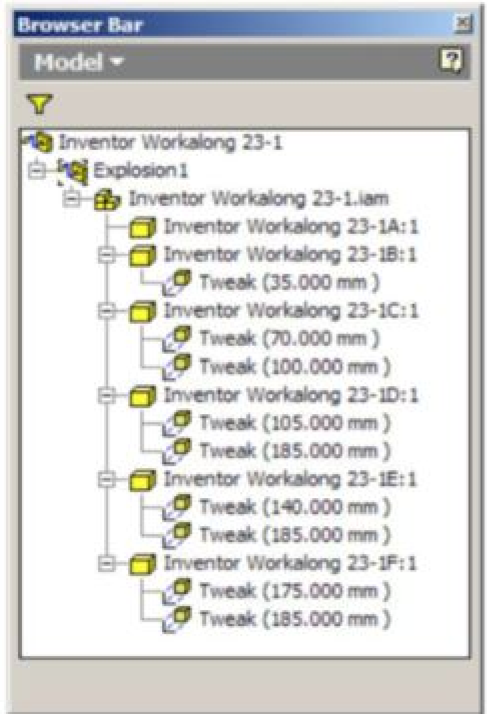


Figure Step 15A

Figure Step 15B

Step 16

Enter the ANIMATE command. It will open the Animation dialogue box. Set the Interval to 25 and the Repetitions to 1. (Figure Step 16)

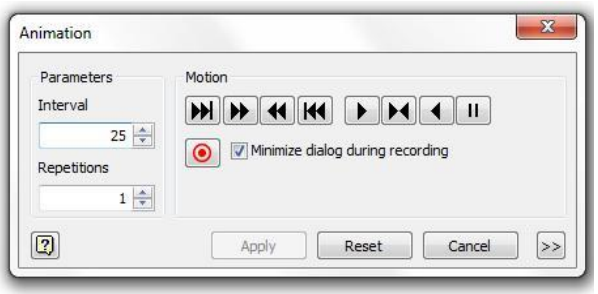


Figure Step 16

Step 17

Study Figure Step 17A, 17B, 17C, and 17D. Play the animation of the assembly that you created in this workalong. Try the different options allowed in the Motion box. (Figure Step 17A, 17B, 17C, and 17D)

Figure Step 17A

Figure Step 17B

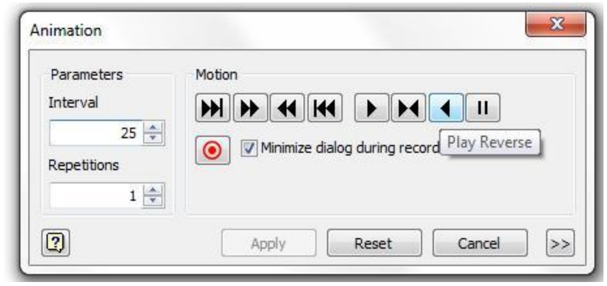


Figure Step 17C

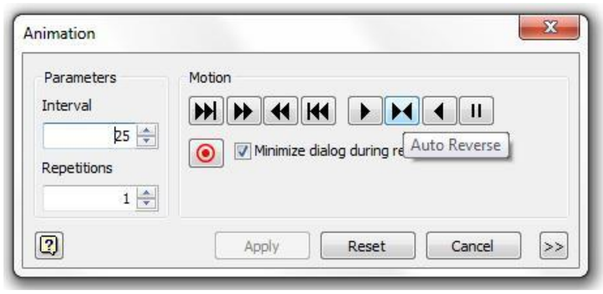


Figure Step 17D

Step 18

Set the Interval to 15 and the Repetitions to 3. Play the animation using these parameters both forward and in reverse. (Figure Step 18)

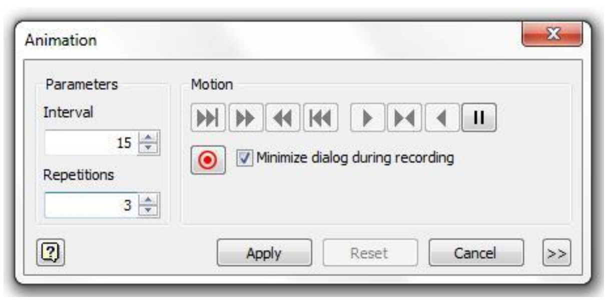
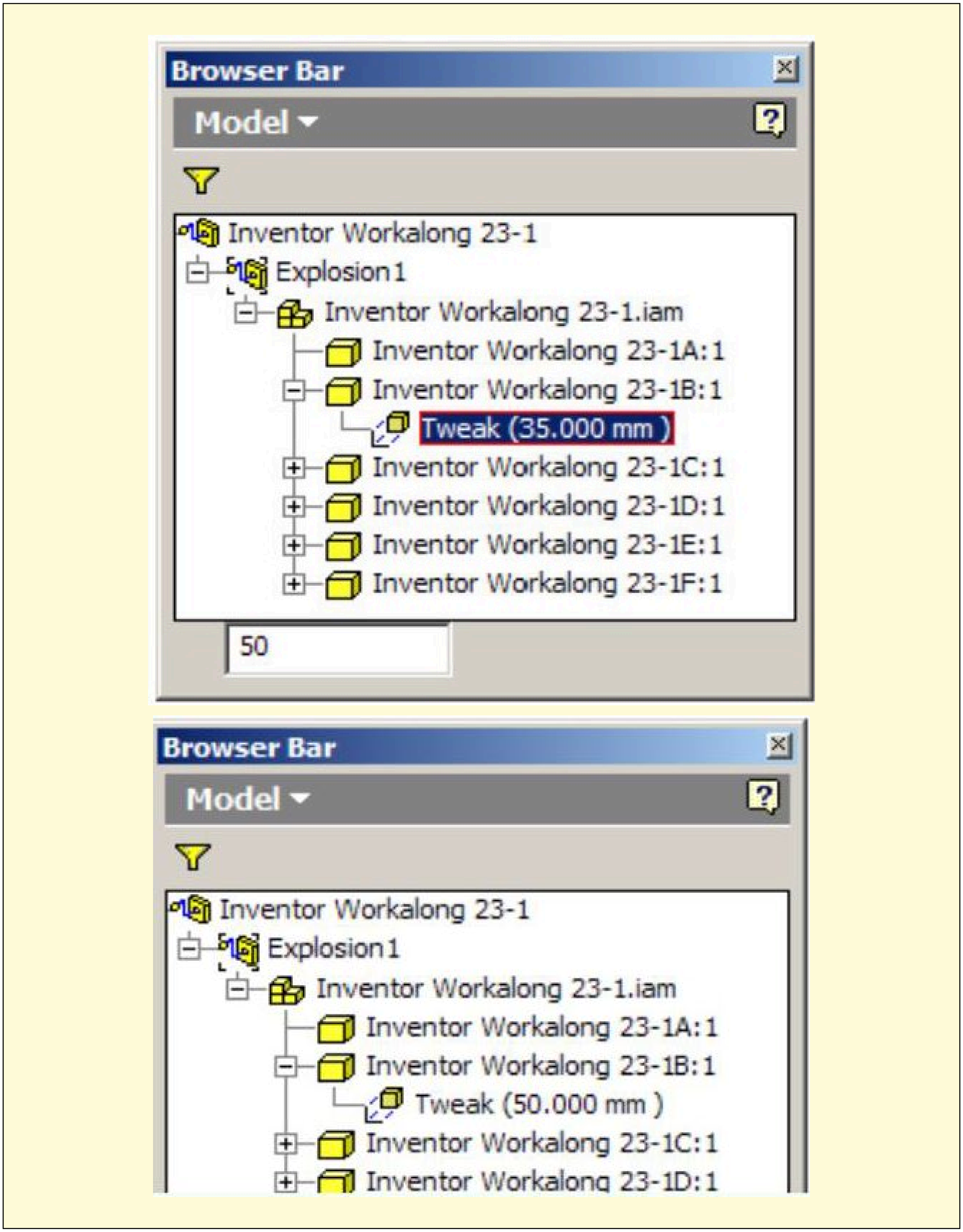


Figure Step 18

Step 19

Save and close the file.

USER TIP: The tweaked distance can be edited in the Browser bar. Locate the tweak you want to change and select it. Once selected, it will highlight and at the bottom of the browser bar you can change the distance as shown in figure below left. The revised tweak will display as shown in the figure below right.


Browser Bar
Model
Inventor Workalong 23-1
Explosion1
Inventor Workalong 23-1.iam
Inventor Workalong 23-1A:1
Inventor Workalong 23-1B:1
Tweak (35.000 mm)
Inventor Workalong 23-1C:1
Inventor Workalong 23-1D:1
Inventor Workalong 23-1E:1
Inventor Workalong 23-1F:1
50
Browser Bar
Model
Inventor Workalong 23-1
Explosion1
Inventor Workalong 23-1.iam
Inventor Workalong 23-1A:1
Inventor Workalong 23-1B:1
Tweak (50.000 mm)
Inventor Workalong 23-1C:1
Inventor Workalong 23-1D:1

Key Principles

Key Principles in Module 23

1. A presentation file is a file in which a view of an assembly is exploded and can be animated. It can be rotated so that is can be viewed from different angles.
2. An exploded view shows the assembly as if it were dismantled and the components of the assembly shown in the order and orientation they fit together to create the assembly.
3. A tweak is the distance that the part is moved from the grounded part while the trails are the lines in an exploded view that show the relationship of the component to the assembly.
4. After the exploded assembly is tweaked, it can animated to show the assembly and the disassembly of the model. An animation is simply a series of frames or pictures of the assembly displayed one frame at a time. The amount of time between frames is called the interval.

Module 24 2D Drawings – Part 1

Learning Outcomes

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

1. Describe 2D drawing files, drawing sheets, and drawing sizes.
2. Describe and apply the commands BASE VIEW, PROJECTED VIEW, SECTION VIEW, and NEW SHEET to create multiview and isometric views of a solid model on a drawing sheet.

Drawing Files

A *drawing file* contains one or more drawing sheets on which 2 dimensional and/or 3 dimensional scaled views of the solid models contained in part, assembly, or presentation files. The views can be created complete with hidden lines or shading. Annotation can be automatically or manually added to the views as required. A typical drawing sheet with an orthographic, section, and isometric view of a model is shown in Figure 24-1. Dimensioning, inserting text, and filling in the titleblock are taught in Module 25. When the drawing is complete, it can be printed or plotted on paper. A drawing file has the file extension .idw. IDW is an acronym for Inventor Drawing.

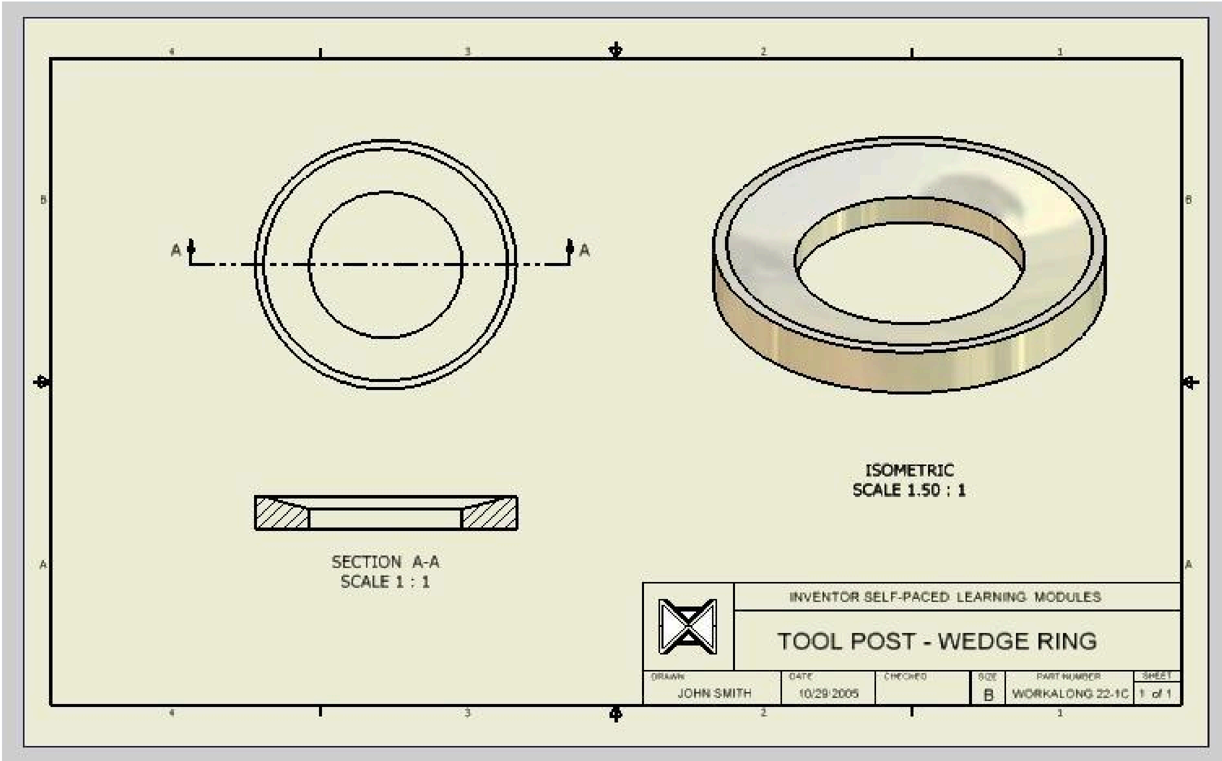


Figure 24-1
A Typical Drawing Sheet in a Drawing File [Click to see image full size]

Model Views

A*model view* is a scaled view orientated at an angle and direction that the solid model or assembly is being viewed and displayed on the drawing sheet. There is no limit to the number of views or the number of solid models from part, assembly, or presentation files that can be placed on a drawing sheet. Views can also be automatically annotated or labeled. There are eight predefined views that you can select from when creating the view. The predefined views are the; base, projection, auxiliary, section, detail, broken, breakout, and overlay.

The Base

A *Base view* is the first view created on the drawing sheet. It controls the scale, orientation, and location of the views projected from it. The orthographic and/or isometric views in the drawing are created from the Base view. For example, if a multiview drawing was being created from a solid model, the Front view is created first as a Base view. The Front view would control the scale and location of the projected Top and Right Side views. See Figure 24-2. To change the scale of all views, only the scale of the Front (Base) view would have to be changed and the Top and Right Side views would automatically change scale to match. If the Front view was moved, the Top and Right Side views would move accordingly to keep their multiview position.

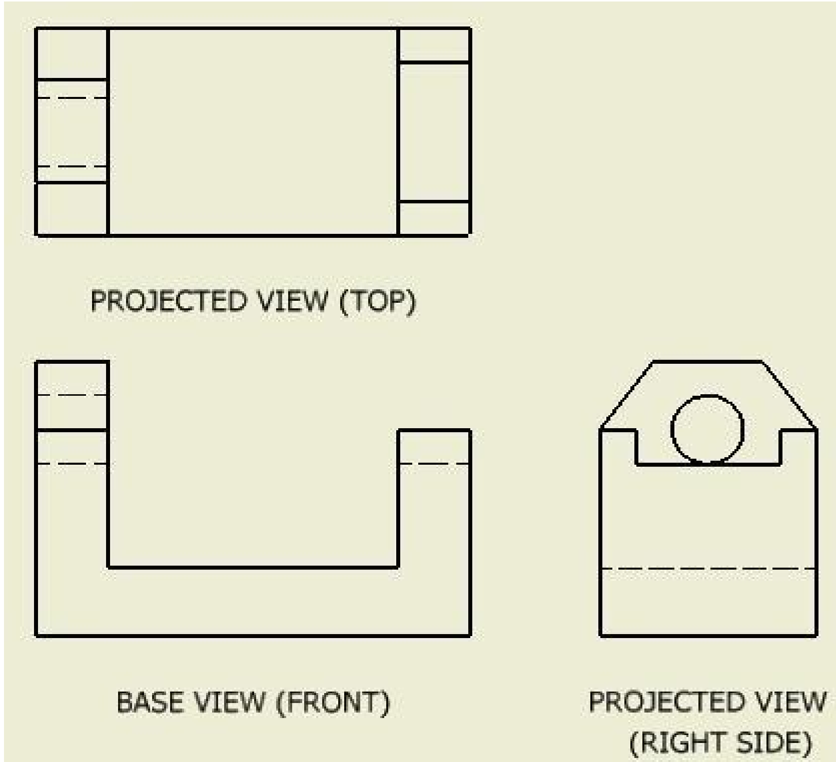


Figure 24-2
Multiview Drawing

Projected Views

A *projected view* is a view projected from a Base view. The scale of a projected view cannot be set since the Base view that you projected it from controls its scale. The Base view also controls the orientation and location of the projected views.

Drawing Sheets

A *drawing sheet* represents a blank piece of paper complete with titleblock and border. The size of drawing sheet can be set by the user. The sheet size can be a custom size set by you or one of the ANSI or ISO drawing sheet standards listed in the table shown in Figure 24-3.

There is no maximum number of drawing sheets that can be created for each drawing file but there must be at least one sheet. Sheets can be created or deleted but Inventor will NOT allow all of them to be deleted since one sheet must exist at all times.

The drawing sheet can be assigned a drawing border and a titleblock which can be created or edited by you. Custom drawing template files containing borders and titleblocks are supplied with the Inventor book.

MUST KNOW: There is no maximum number of drawing sheets that can be created for each drawing file but there must be at least one sheet. One sheet must exist in each drawing file at all times.

ANSI and ISO Standard Drawing Sheet Sizes

ANSI and ISO Standard Drawing Sheet Sizes			
ANSI - English		ISO - Metric	
Size A	81/2 X 11 in	Size A4	210 X 297 mm
Size B	11 X 17 in	Size A3	297 X 410 mm
Size C	17 X 22 in	Size A2	410 X 594 mm
Size D	22 X 34 in	Size A1	594 X 820 mm
Size E	34 X 44 in	Size A0	820 X 1198 mm

Figure 24-3
Standard ANSI and ISO Drawing Sheet Sizes

View Style

A*view style* can be displayed in one of three different styles. The three styles are *hidden line*, *hidden line removed*, and *shaded* as shown in Figure 24-4. The style of a view can be changed as required after the view has been placed.

View Scale

The *scale* of the view is a factor of the number that it is set to. For example, if the scale is set to 1 then the factor of 1 X 1 = 1, full scale or 1:1. If the scale is set to 2 then 2 X 1 = 2 or a scale of 2:1 which is twice the size of the original model. On the other hand, if the scale factor is set to 0.5 then 0.5 X 1 = 0.5 or the scale of 1:2 which would display the view one-half the size of the original model.

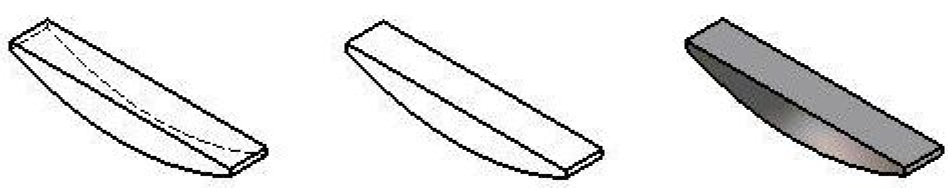


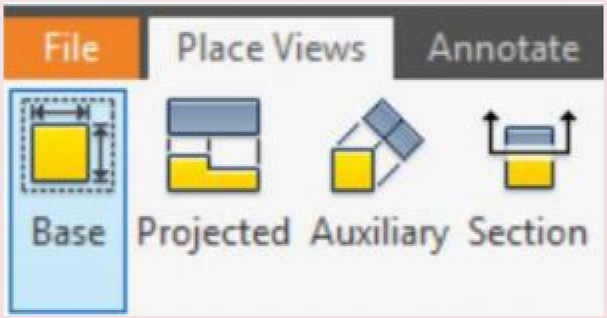
Figure 24-3
View Styles

MUST KNOW: A drawing file contains one or more 2 dimensional drawing sheets on which a 2D and/or 3D scaled views of solid models contained in part, assembly, and presentation files. A drawing file has the file extension .idw. IDW is an acronym for Inventor Drawing.

Inventor Command: BASE VIEW

The BASE VIEW command is used to create a Base view of the solid model contained in a part, assembly, or presentation file on the drawing sheet. The scale, style, labeling, and orientation of the view can be set when the view is created.

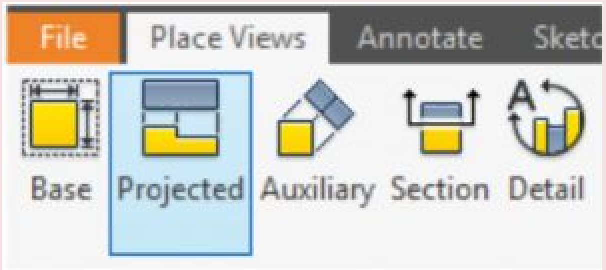
Shortcut: **none**



Inventor Command: PROJECTED VIEW

The PROJECTED VIEW command is used to create a projected view from a Base view. The style, labeling, and direction from the Base view can be set by you when the view is created.

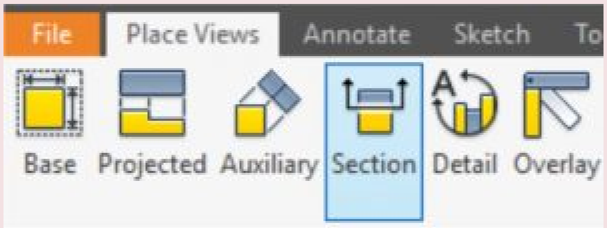
Shortcut: **none**



Inventor Command: SECTION VIEW

The SECTION VIEW command is used to create a Section view from an existing view. The location, style, labeling, and direction from the selected view can be set by you when the view is created.

Shortcut: **none**



USER TIP: ANSI is an acronym for American National Standard Institute. ANSI has set drawing standards that are widely adapted and followed by most companies working in English or Imperial measurements. To read more about ANSI see: http://www.ansi.org/
ISO is an acronym for International Organization for Standardization. ISO has set the drawing standards that are widely adapted and followed by most companies working in Metric measurements. To read more about ISO see: http://www.iso.org/iso/en/ISOOnline.frontpage

WORK ALONG: Creating 2D Drawings

Step 1

Check the default project and if necessary, set it to: Inventor Course.

Step 2

Enter the NEW command to start a new drawing file. Enable the English tab and select the template: Modules Drawing ANSI (in).idw. (Figure Step 2)

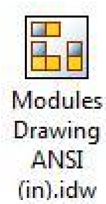


Figure Step 2

AUTHOR'S COMMENTS: See the User Tip before WORKALONG: Creating 2D Drawings for a explanation of the acronyms ANSI and ISO.

Step 3

When the drawing file is opened, it will display an A size drawing complete with border and titleblock. Save the file with the name: Inventor Workalong 24-1. (Figure Step 3)

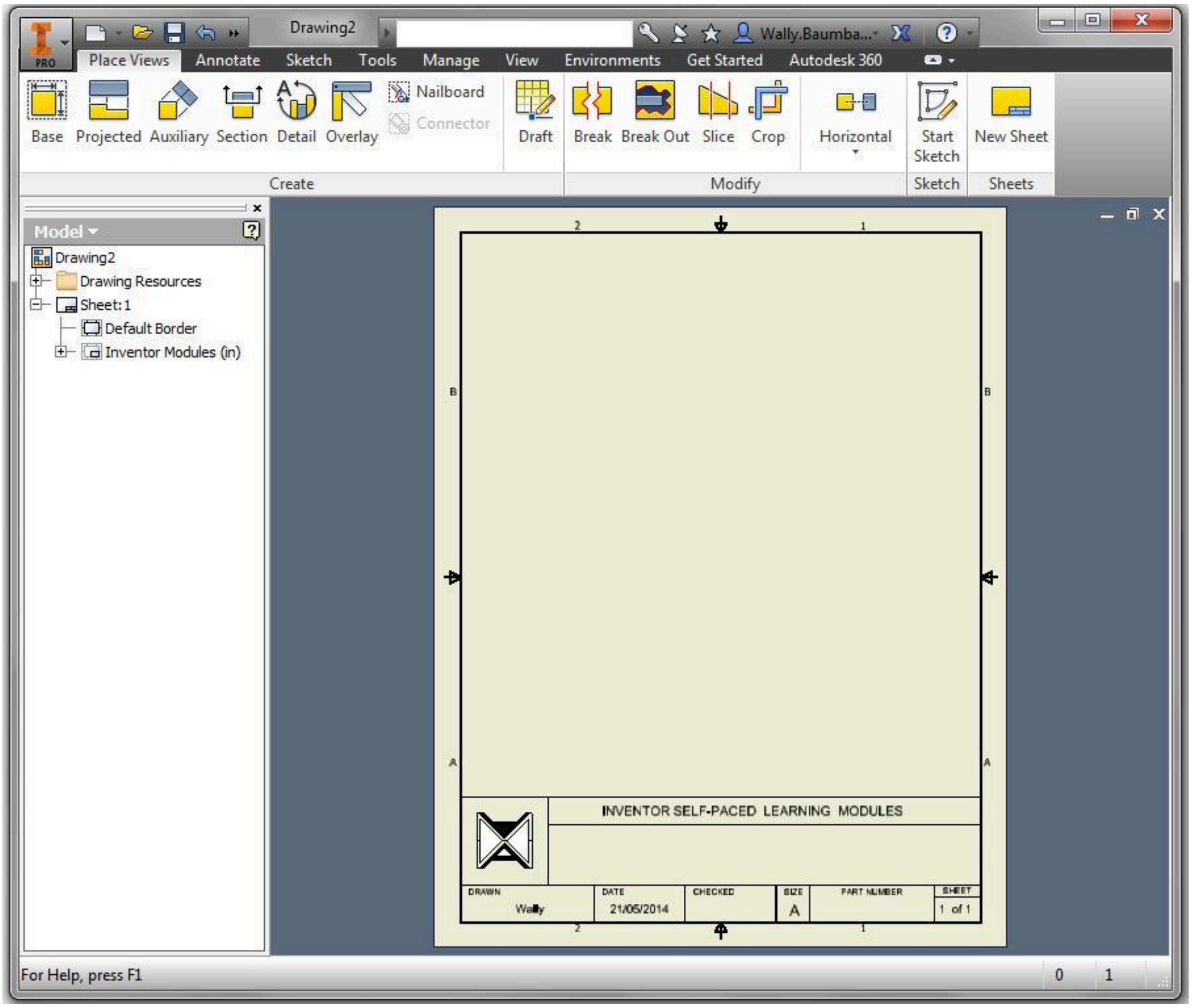


Figure Step 3 [Click to see image full size]

Step 4

Enter the BASE VIEW command to create the Base view. It will open the Drawing View dialogue box. Set the Orientation (view) to Front and the Style to Hidden Line. Ensure that the dialogue box matches the figure. (Figure Step 4)

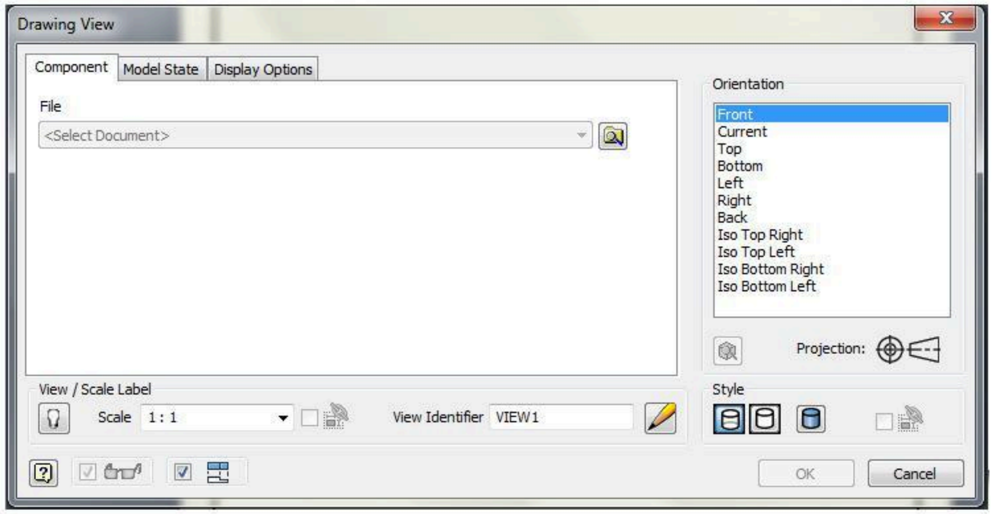


Figure Step 4 [Click to see image full size]

Step 5

Click OK. In the Open dialogue box, select the part: <u>Inventor Workalong 22-1D.ipt</u>. (Figure Step 5)

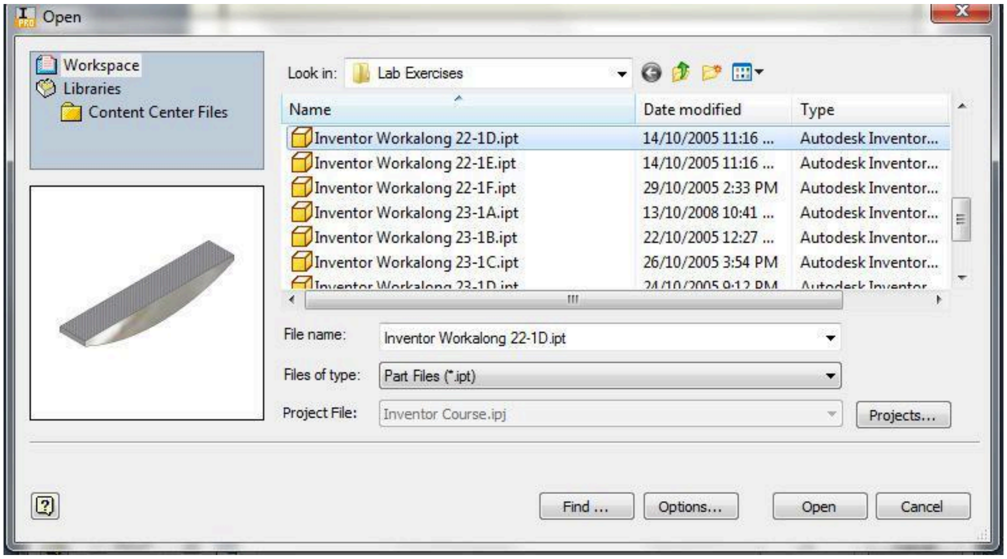


Figure Step 5 [Click to see image full size]

Step 6

Select the location for the Front view. Don't be too concerned where you locate it since it can be moved later. Try to locate it close to where it is shown in the figure. (Figure Step 6A and 6B)

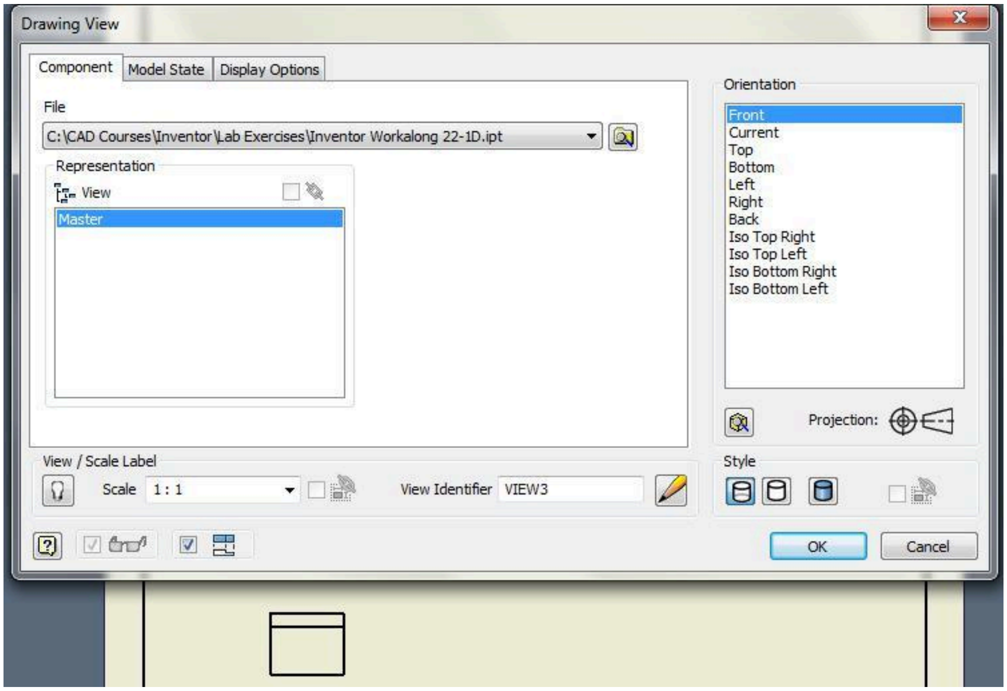


Figure Step 6A [Click to see image full size]

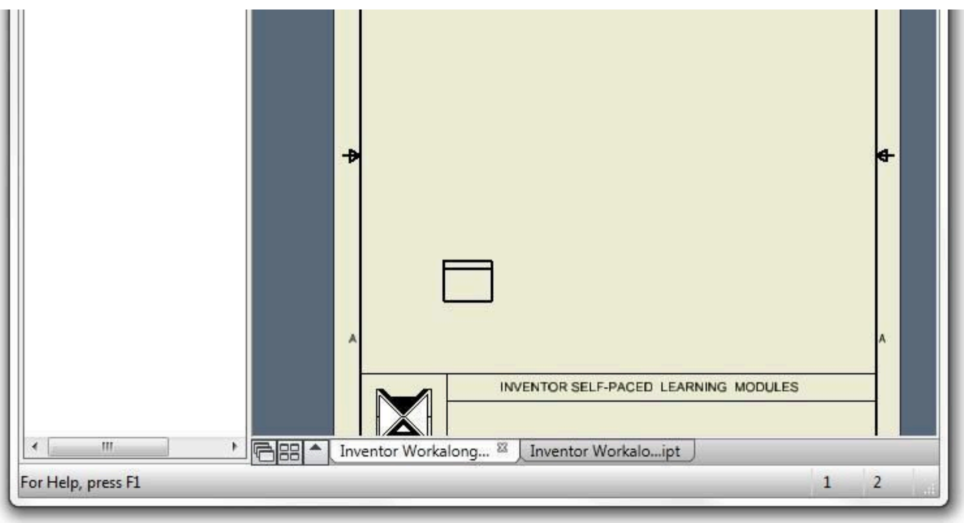


Figure Step 6B [Click to see image full size]

Step 7

Enter the PROJECT VIEW command and select the Base view as the view to project from. (Figure Step 7)

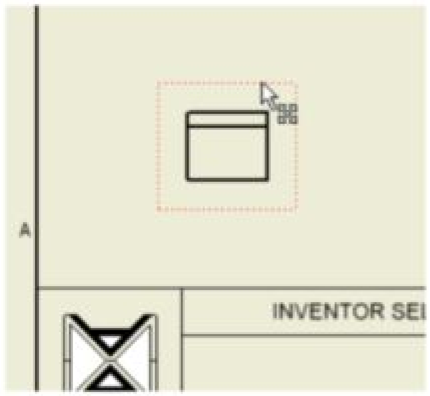


Figure Step 7

Step 8

Move the cursor up to locate the Top view. Click the mouse at the desired location. Right click the mouse. In the Right-click menu, select Create. (Figure Step 8A and 8B)

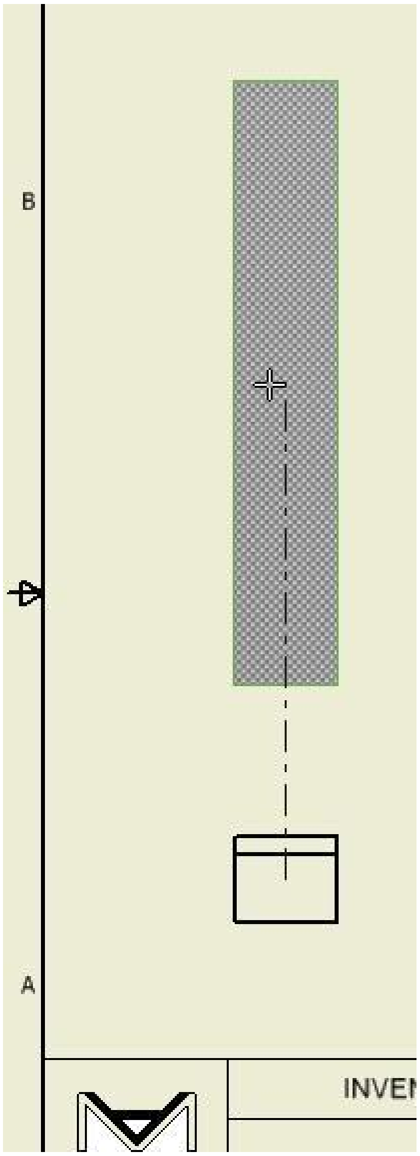


Figure Step 8A

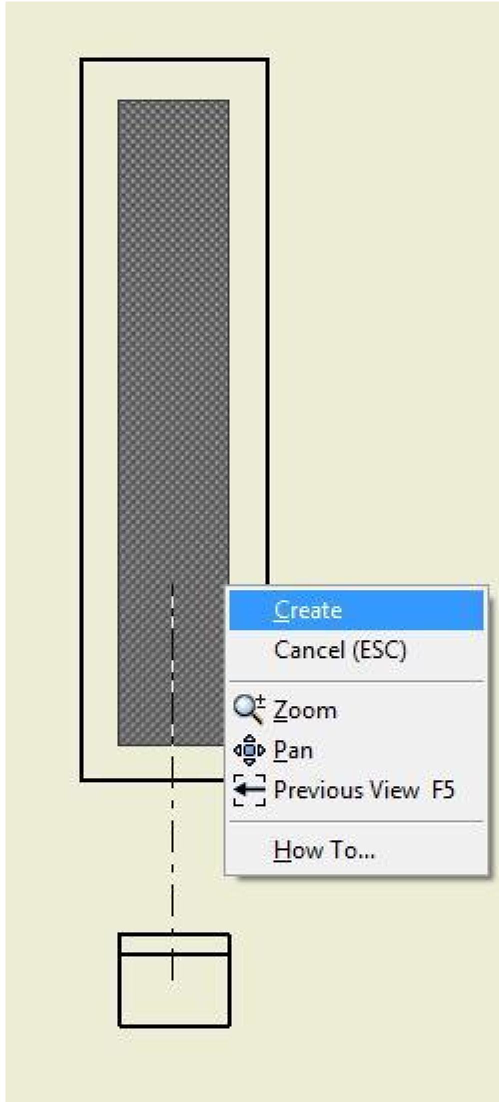


Figure Step 8B

Step 9

Using what you just learned, use the PROJECT VIEW command to create the Right Side view. (Figure Step 9)

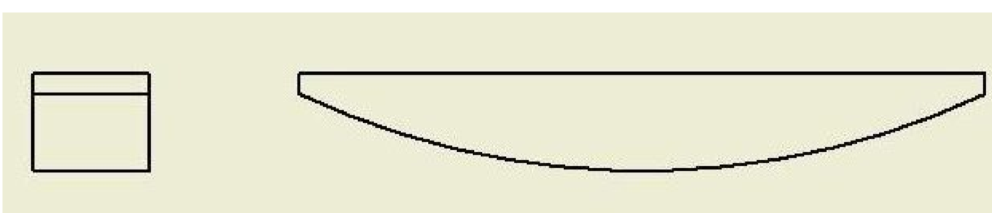


Figure Step 9

Step 10

Your drawing should now contain the Top, Front and Right Side views of the part. (Figure Step 10)

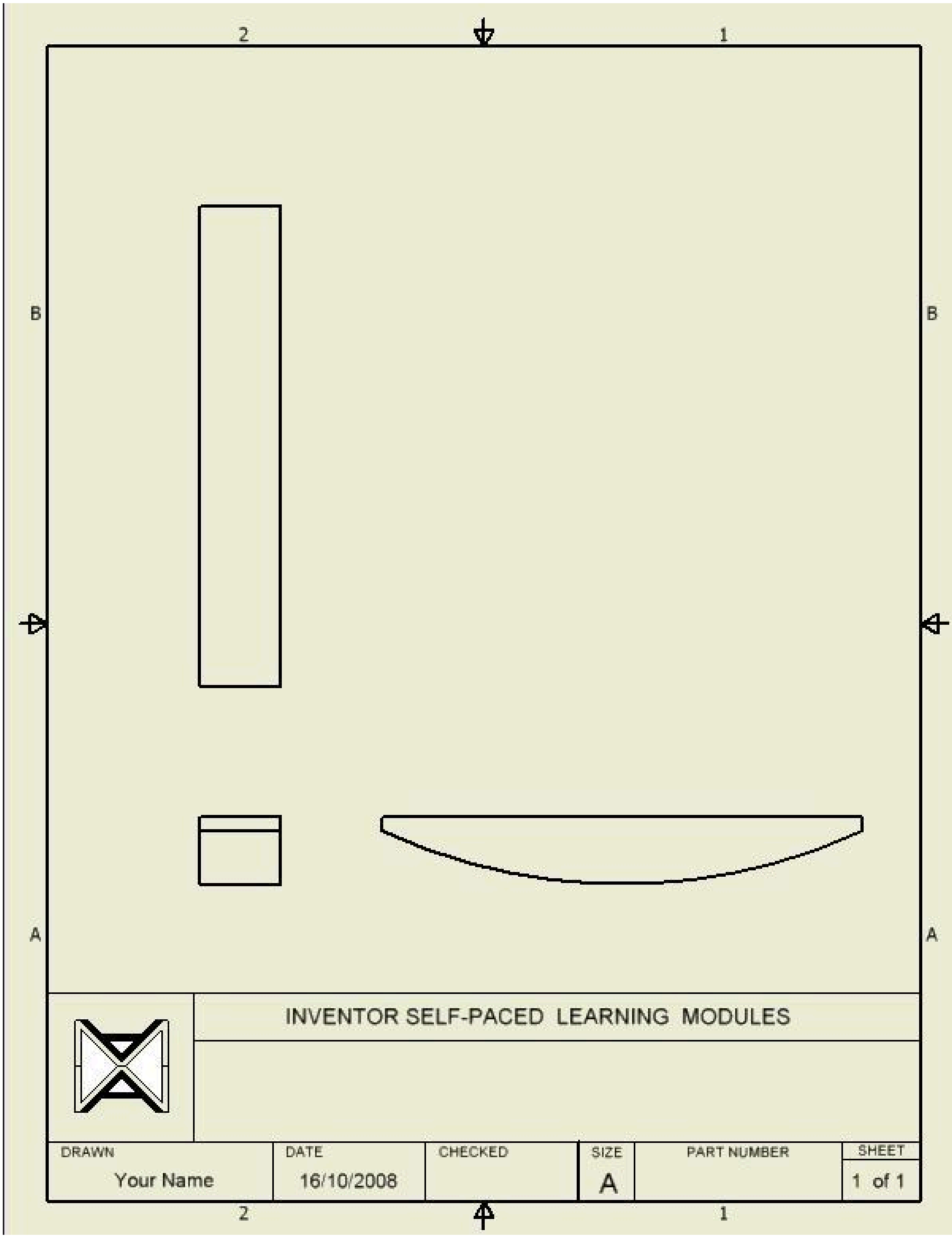


Figure Step 10 [Click to see image full size]

Step 11

Enter the PROJECT VIEW command. Select the Base view (Front view) and project an Isometric view from it. (Figure Step 11)

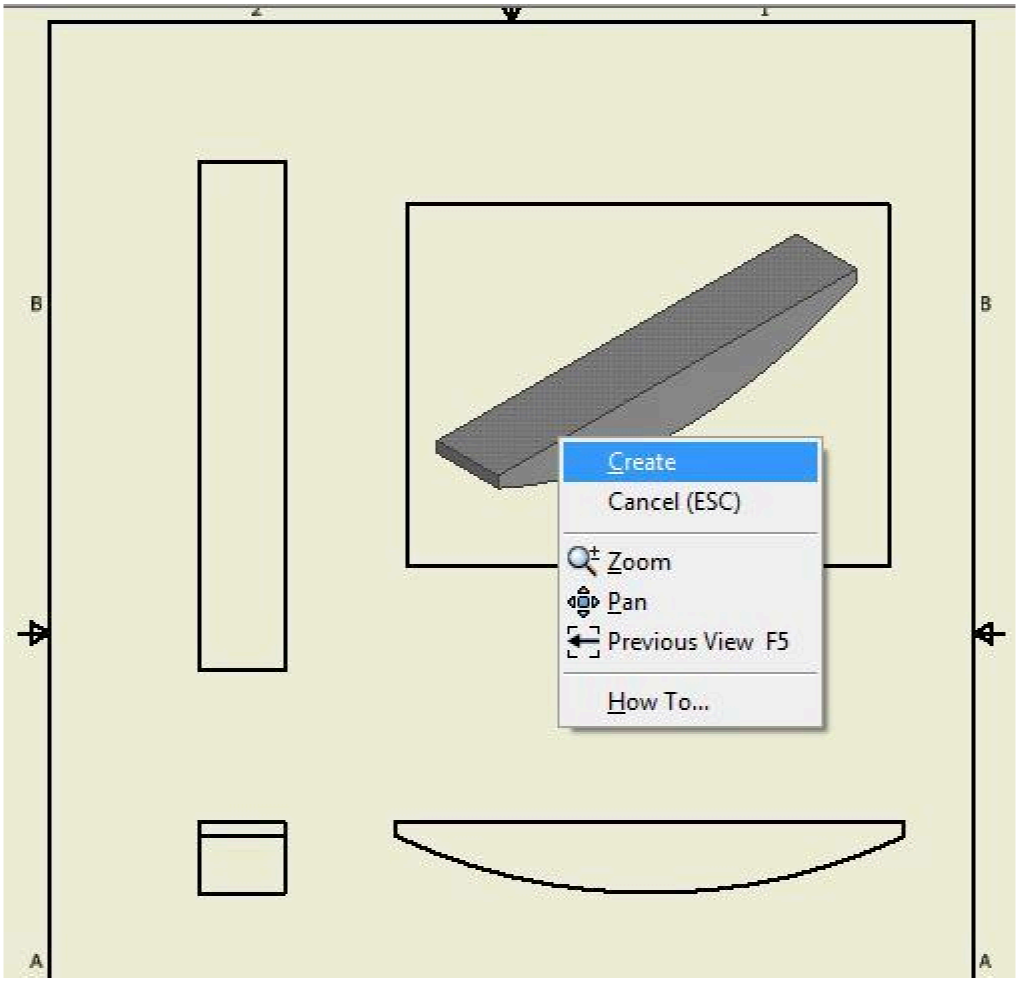


Figure Step 11

Step 12

Right click the Isometric view. In the Right-click menu, click Edit View. (Figure Step 12)

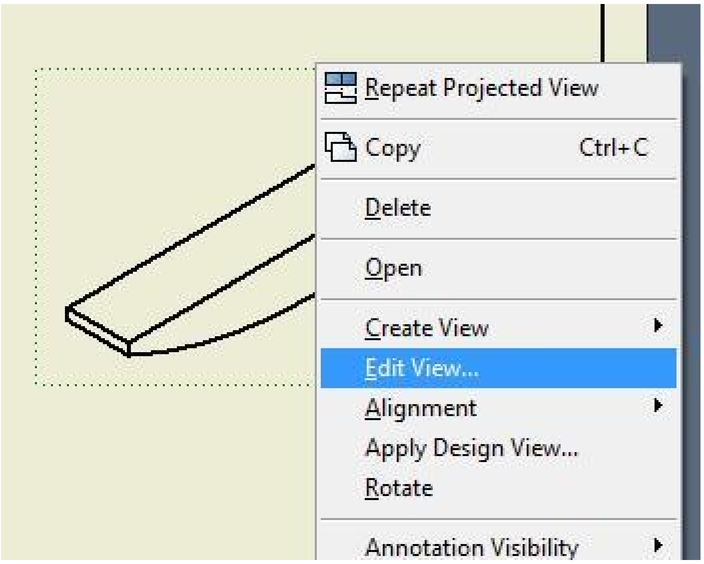


Figure Step 12

Step 13

In the Edit View dialogue box enable the display View/Scale label (turn the light bulb on). Set the Scale to 0.75:1, View Identifier to ISOMETRIC and the Style to Shaded. (Figure Step 13A and 13B)

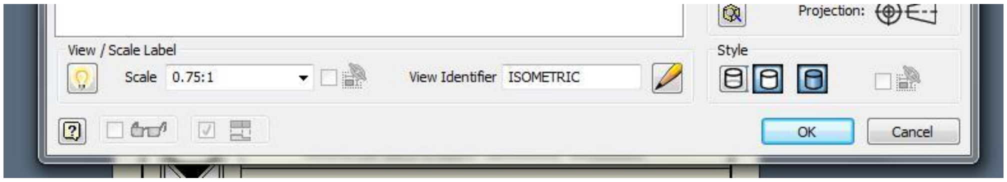


Figure Step 13A [Click to see image full size]

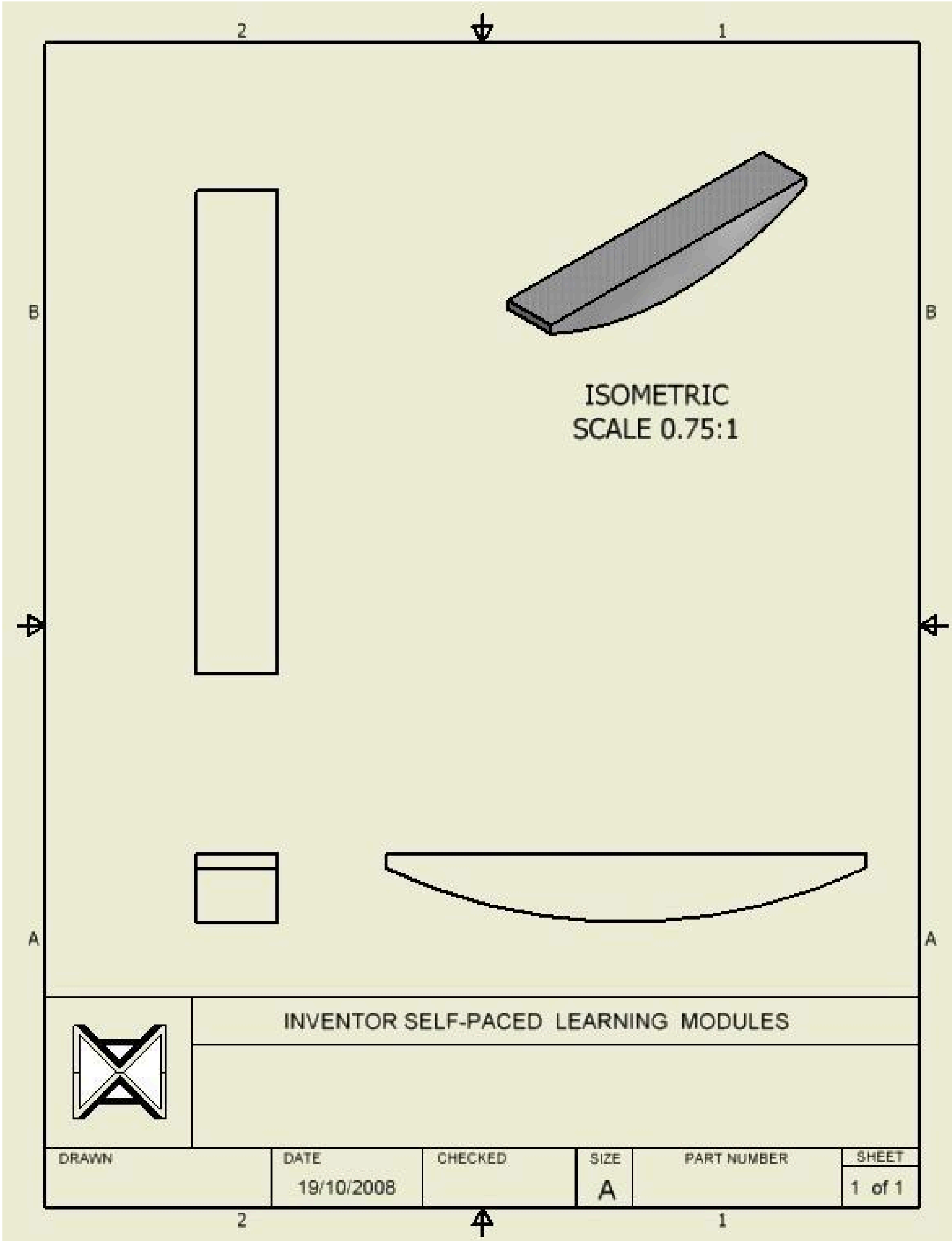


Figure Step 13B [Click to see image full size]

Step 14

Right click the file name in the Browser bar. In the Right-click menu, click New Sheet. An A size drawing sheet will display in the Graphic window. The new sheet will be labeled Sheet:2 and will display in the Browser bar. (Figure Step 14)

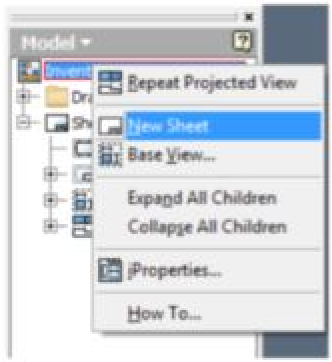


Figure Step 14

Step 15

Right-click Sheet:2. In the Right-click menu, click Edit Sheet. In the Edit Sheet dialogue box, pull down the Size list and select B to change Sheet 2 to a B size. Ensure that Landscape is enable. (Figure 15A, 15B, and 15C)

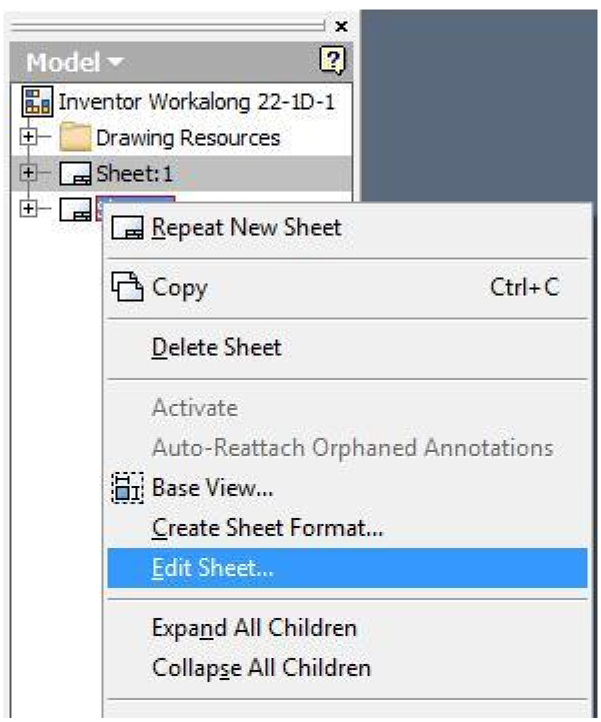


Figure Step 15A

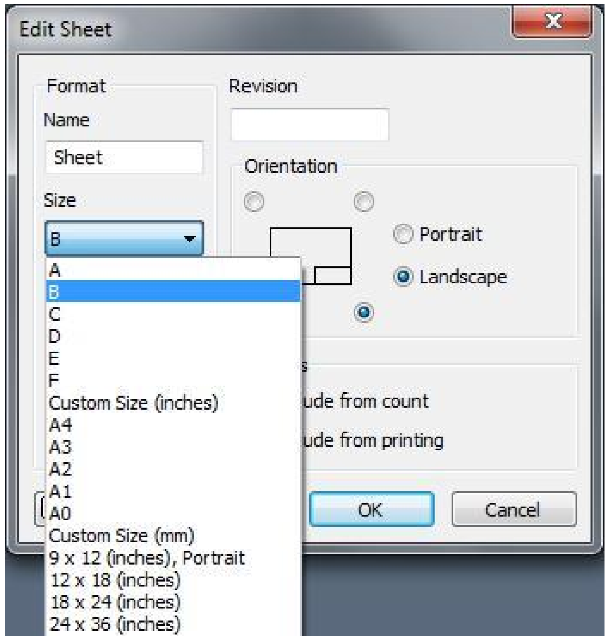


Figure Step 15B

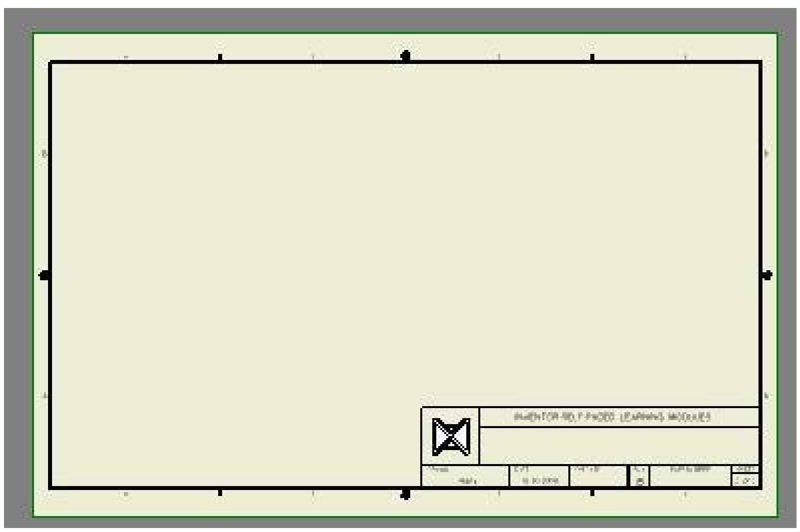


Figure Step 15C [Click to see image full size]

Step 16

Using what you just learned, create a Base view of Top view of part: Inventor Workalong 22-1C.ipt that you created in Module 22. (Figure Step 16)

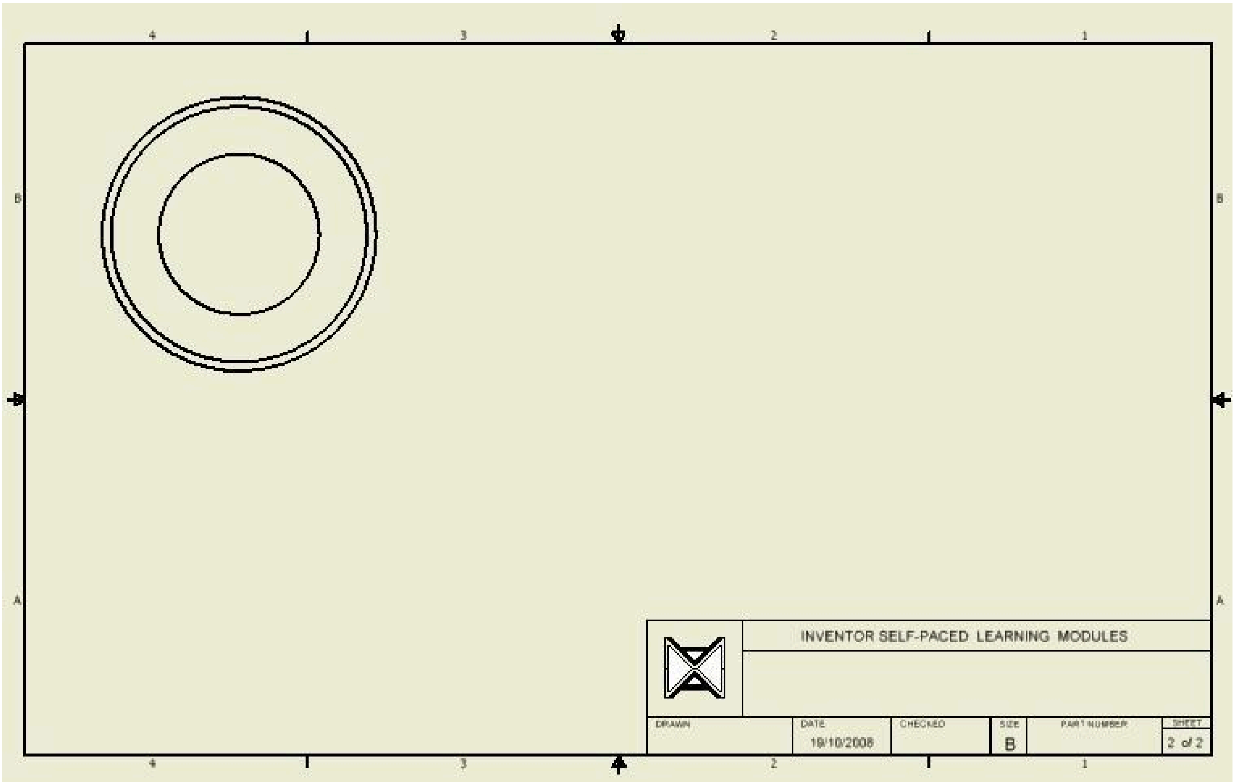


Figure Step 16 [Click to see image full size]

Step 17

Enter the SECTION VIEW command. Move the cursor to the centre of the circle on the Top view until it displays the green snap circle. You may have to move the cursor touching the circle circumference and then move back to the centre. Do NOT select the green snap circle, wait until it displays. Move the cursor to the right and you will see a dashed line indicating an implied line which is orthographic or horizontal, in this case. move outside the view. The yellow grid snap circle will display. Select a location when the grid snap is displayed. (Figure Step 17A and 17B)

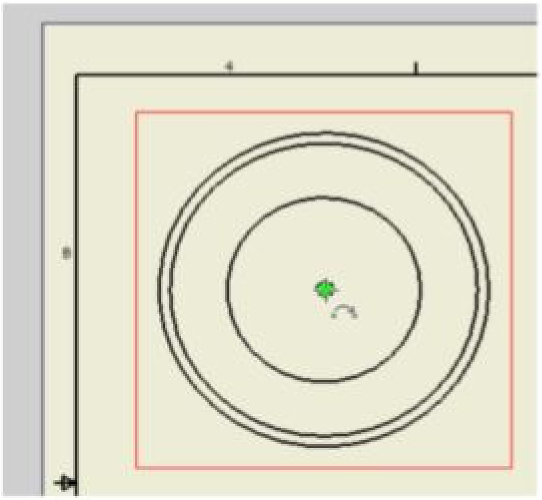


Figure Step 17A

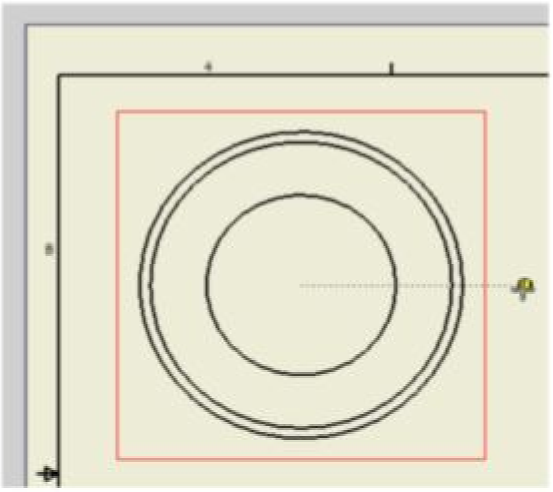


Figure Step 17B

Step 18

Move the cursor across to the other side of the view and when the geometry constraint (horizontal) displays, select a location about the same distance from the view as you did for the other side of the view. Right-click the mouse. In the Right-click menu, select Continue. (Figure Step 18A and 18B)

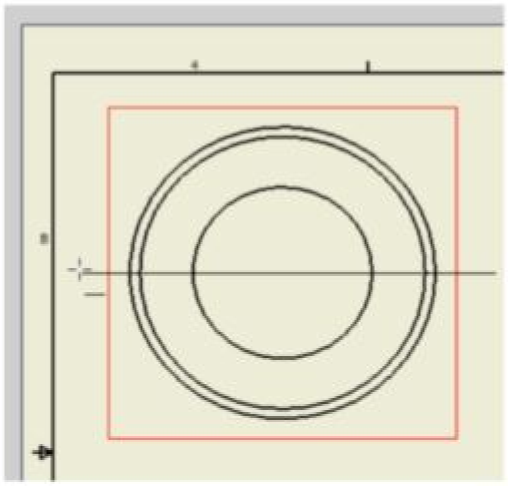


Figure 18A

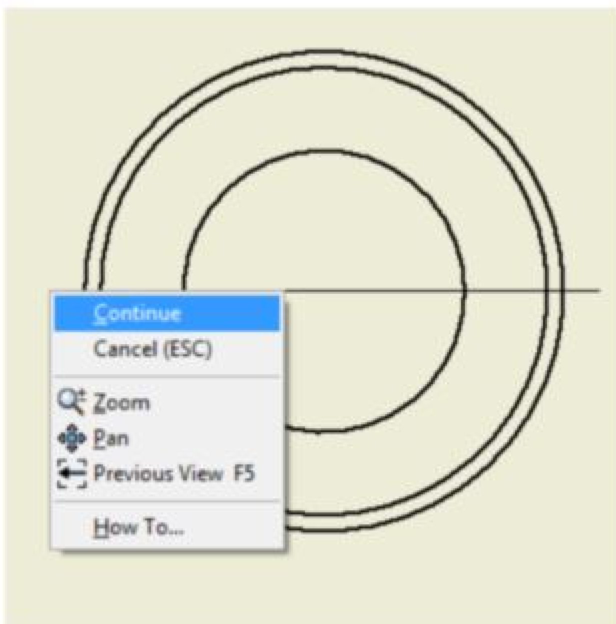


Figure 18B

Step 19

Move the cursor down and the section view will display. The Section View dialogue box will open. Set the dialogue box as shown in the figure. (Figure Step 19A and 19B)

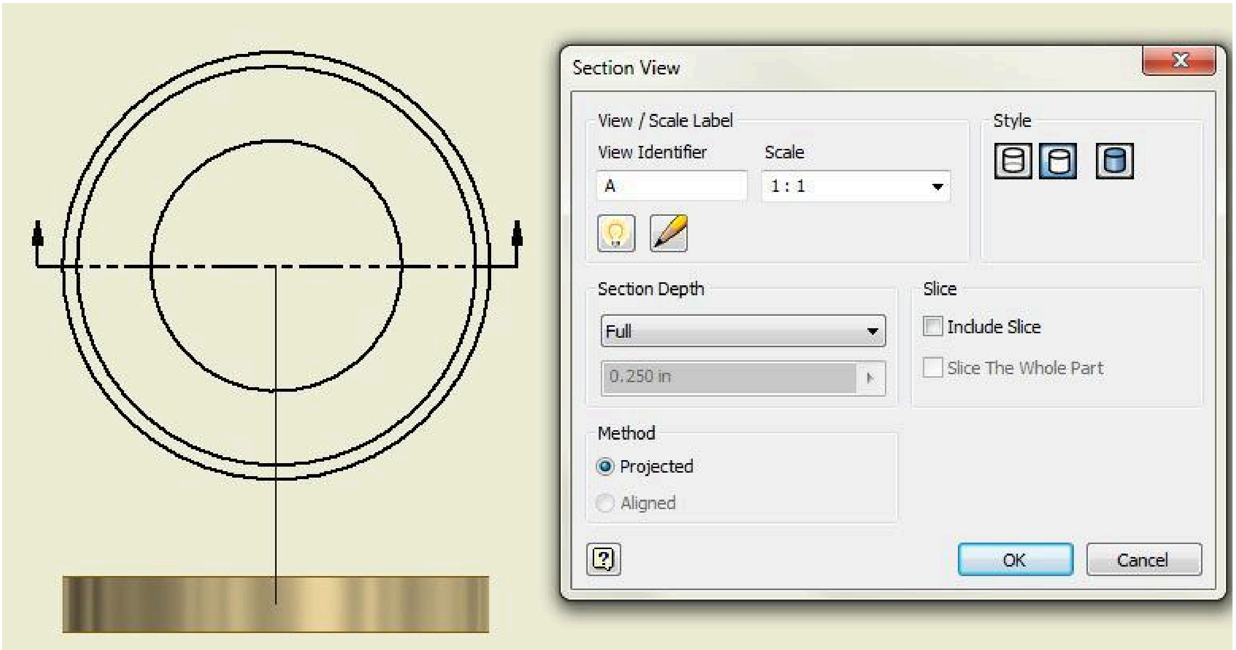


Figure 19A [Click to see image full size]

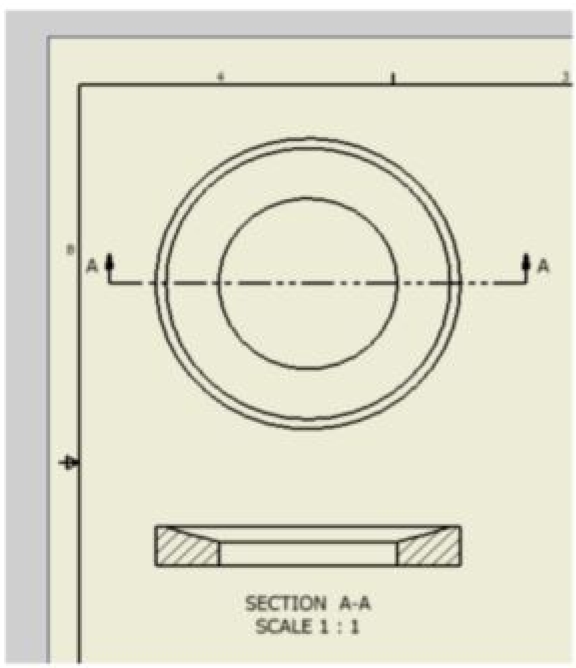


Figure 19B

Step 20

Using the BASE VIEW command, create an Isometric view and set the scale to 1.25:1. (Figure Step 20)

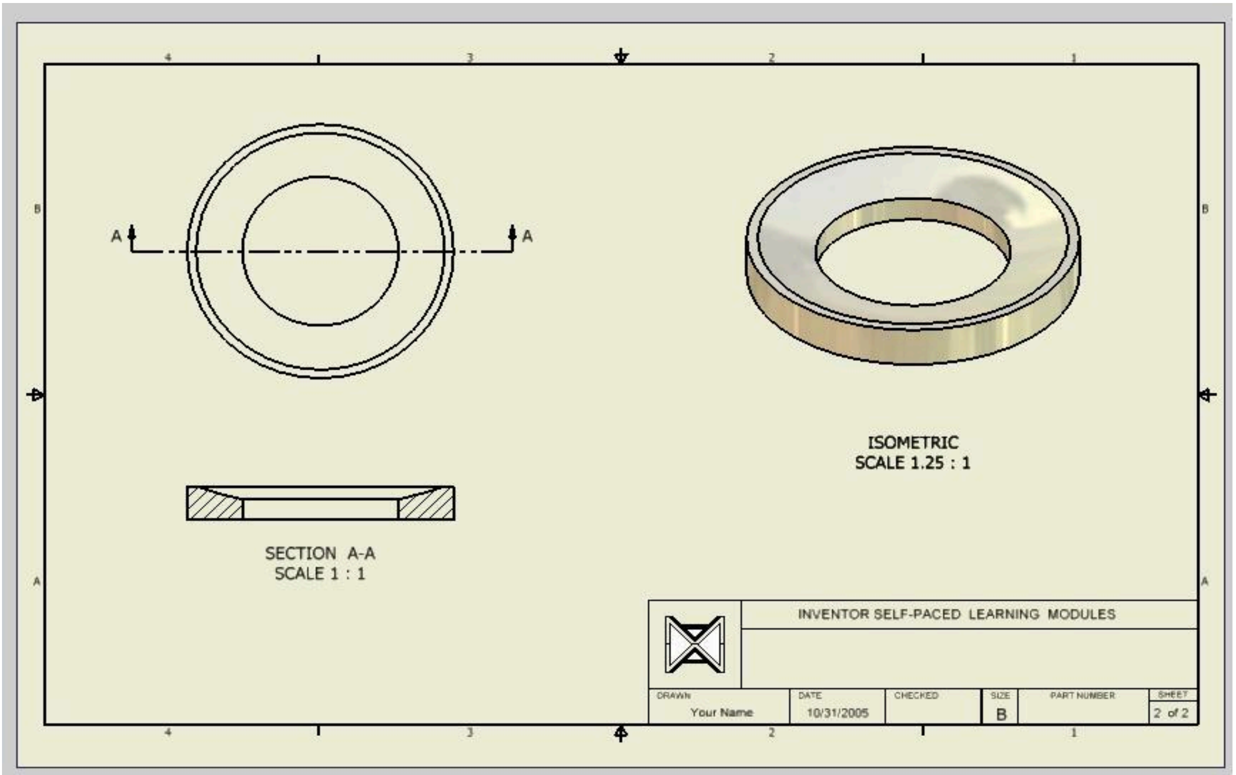
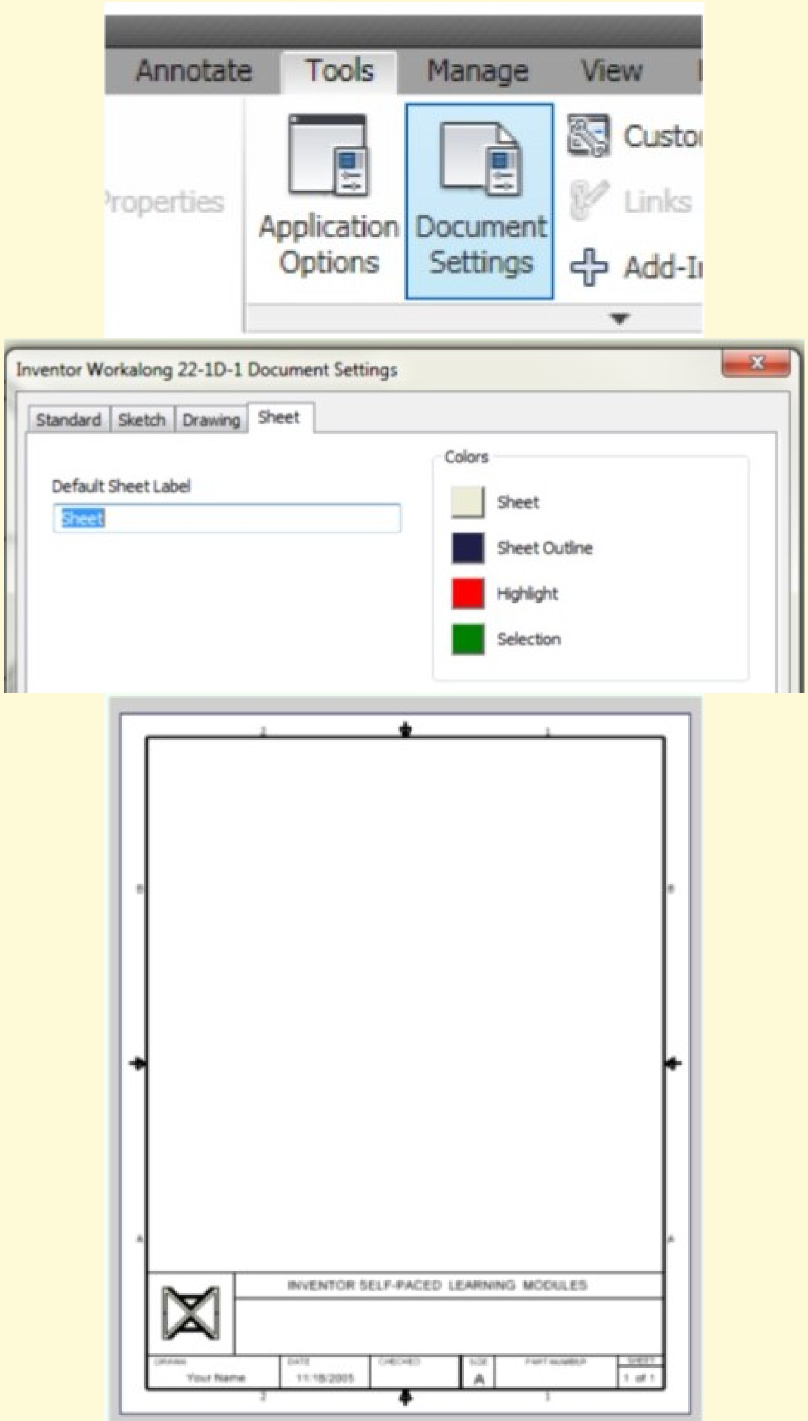


Figure Step 20 [Click to see image full size]

Step 21

Save and close the drawing file.

USER TIP: The default labels and colors can be configured for each drawing file. Originally, these setting came from the template file that you used to start the drawing file. Any changes that you make will only affect the current drawing file. Enter the DOCUMENT SETTINGS command. See figure right. To change the default labels of the sheets, simply edit the value as shown in the figure below. To change the background colour of the Sheet, the Sheet Outline, Highlight or Selection, select the one you want from the Color dialogue box. In this example, the colour of the sheet was change to white.


Annotate
Tools
Manage
View
Application Options
Document Settings
Links
Inventor Workalong 22-1D-1 Document Settings
Standard
Sketch
Drawing
Sheet
Default Sheet Label
Sheet
Colors
Sheet
Sheet Outline
Highlight
Selection
INVENTOR SELF-PACED LEARNING MODULES
A

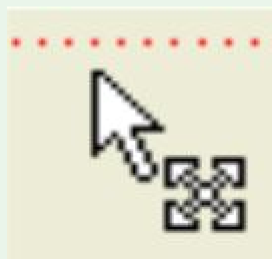
USER TIP: To move a view on the drawing sheet, move the cursor onto the view's border. When you are on the border, it will highlight as shown in Step 1. If the view is a base view or a projected view that has a view dependent on it, the Graphic cursor will display as shown on the left. If the view is a Base view or a projected view that does NOT have view dependent on it, the cursor will display as shown on the right. To move a view, simply press and hold down the left mouse button and drag it to desired location as shown in Step 2.

Key Principles

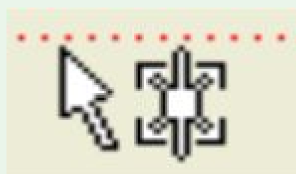
Key Principles in Module 24

1. A drawing file contains one or more 2 dimensional drawing sheets on which a 2D and/or 3D scaled views of solid models contained in part, assembly, and presentation files. A drawing file has the file extension .idw. IDW is an acronym for Inventor Drawing.
2. A Base view is the first view created by the user. It controls the scale, orientation, and location of the orthographic views projected from it.

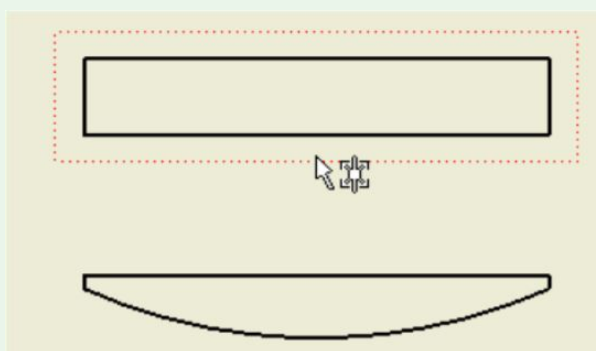
3. A projected view is a view projected from a Base view. The scale of a projected view cannot be set since the Base view that you projected it from controls its scale. The Base view also controls the orientation and location of the projected views.
4. A drawing sheet represents a blank piece of paper complete with titleblock and border. The size of drawing sheet can be set by the user. The sheet size can be a custom size set by the user or one of the ANSI or ISO drawing sheet standards.
5. There is no maximum number of drawing sheets that can be created for each drawing file but there must be at least one sheet. One sheet must exist in each drawing file at all times.
6. A view style can be displayed in one of three different styles. The three styles are hidden line, hidden line removed, and shaded. The style of a view can be changed as required after the view has been placed.
7. The scale of the view is a factor of the number that it is set to.



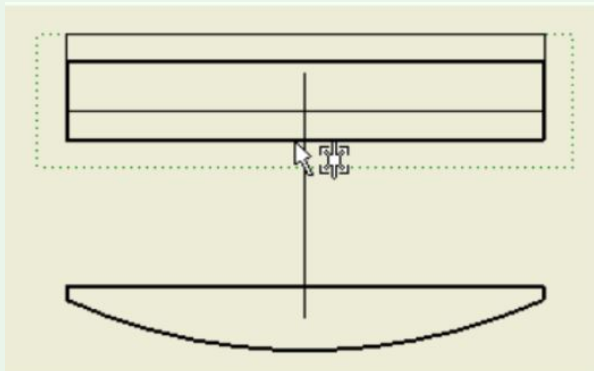
Move



Project



Step 1



Step 2

Lab Exercise 24-1

Time allowed: 2 hours.

Part Name	Project	Units	Template	Color	Material
See Below	Inventor Course	Inches	See Below	N/A	N/A

Step 1

Create the following drawings and ensure the following:
A There is a separate file for each drawing sheet.
B Create the same views as shown.
C If the scale is not indicated, set it to full scale or 1:1.
D Each drawing file has only one drawing sheet.
E Save the drawing files with the Drawing Name shown for each part.

Part: Base
Drawing Size: B
Drawing Name: Inventor Lab 24-1A.idw Part Name: Inventor Workalong 22-1A.ipt

Template: English-Modules Drawing ANSI (in).idw (Figure Step 1A)

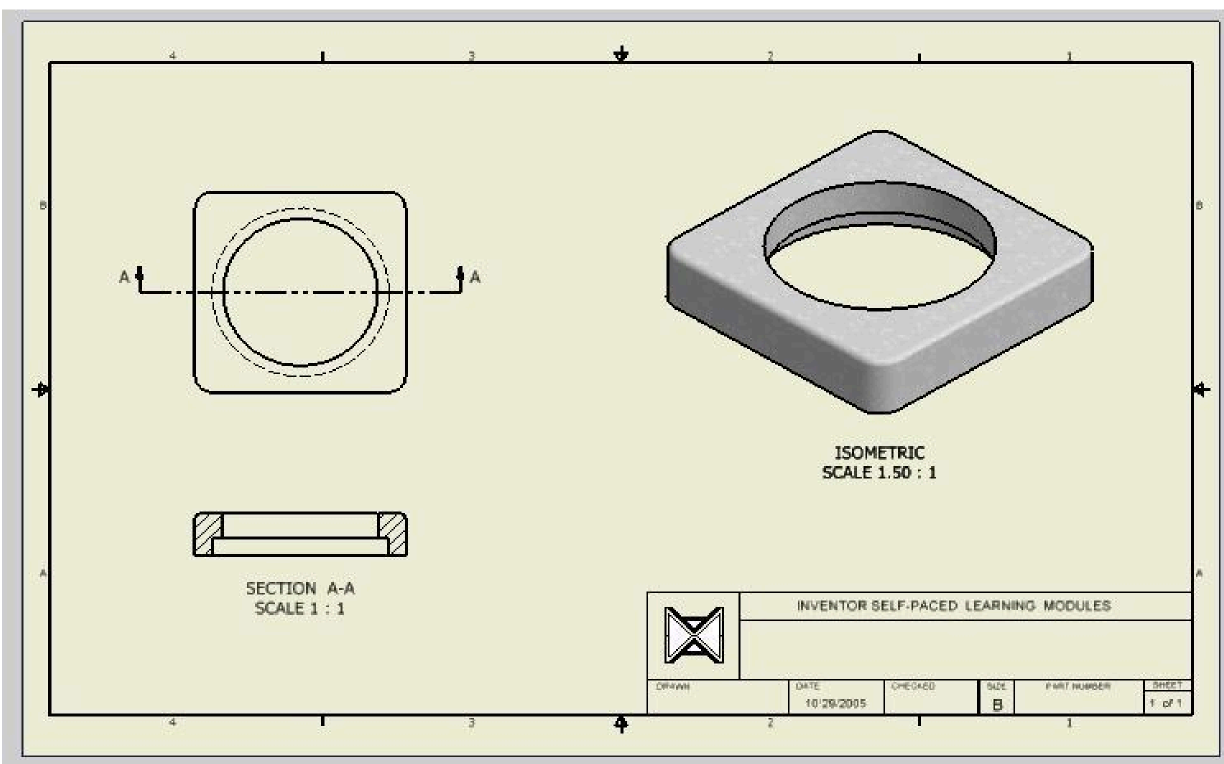


Figure Step 1A [Click to see image full size]

Part: Post
Drawing Size: C
Drawing Name: Inventor Lab 24-1B.idw
Part Name: Inventor Workalong 22-1B.ipt
Template: English-Modules Drawing ANSI (in).idw (Figure Step 1B)

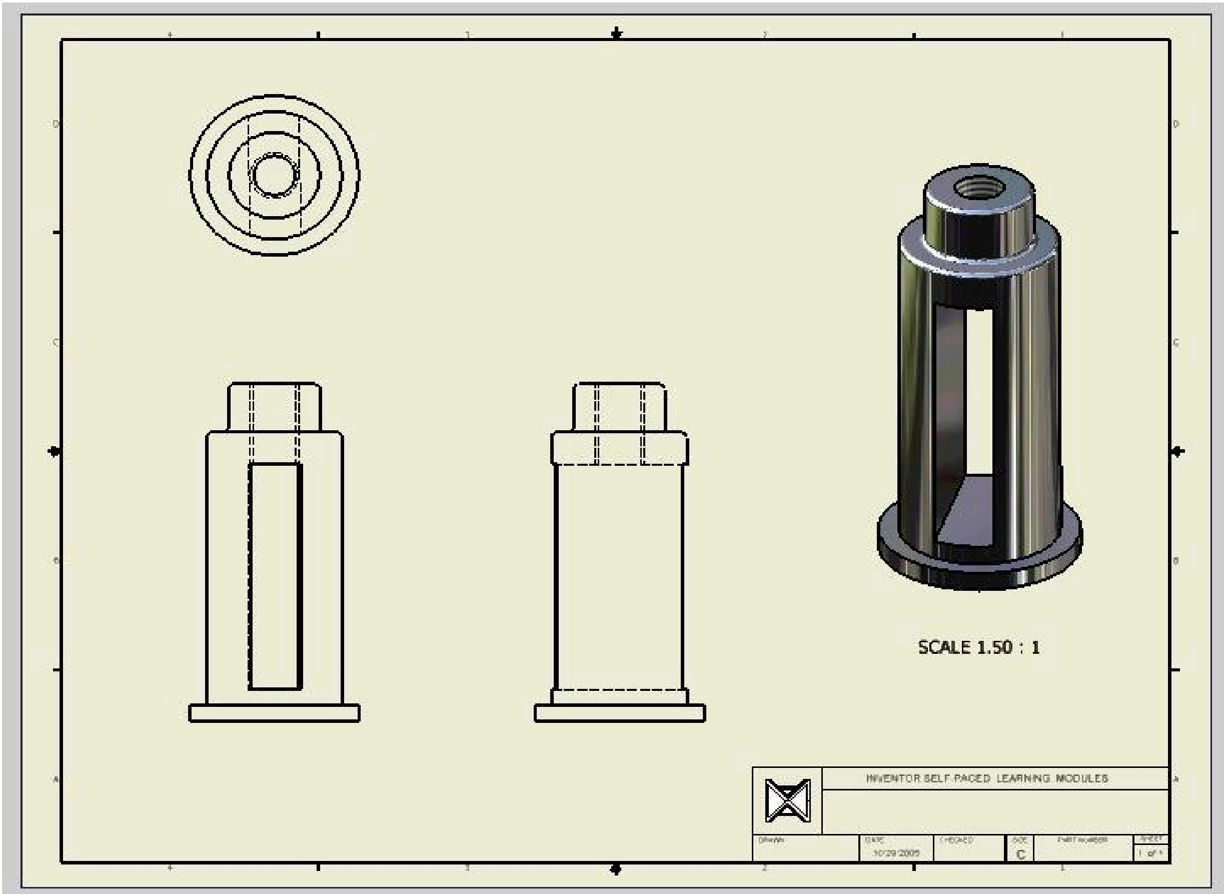


Figure Step 1B [Click to see image full size]

Part: Screw
Drawing Size: A
Drawing Name: Inventor Lab 24-1C.idw
Part Name: Inventor Workalong 22-1E.ipt
Template: English-Modules D

rawing ANSI (in).idw (Figure Step 1C)

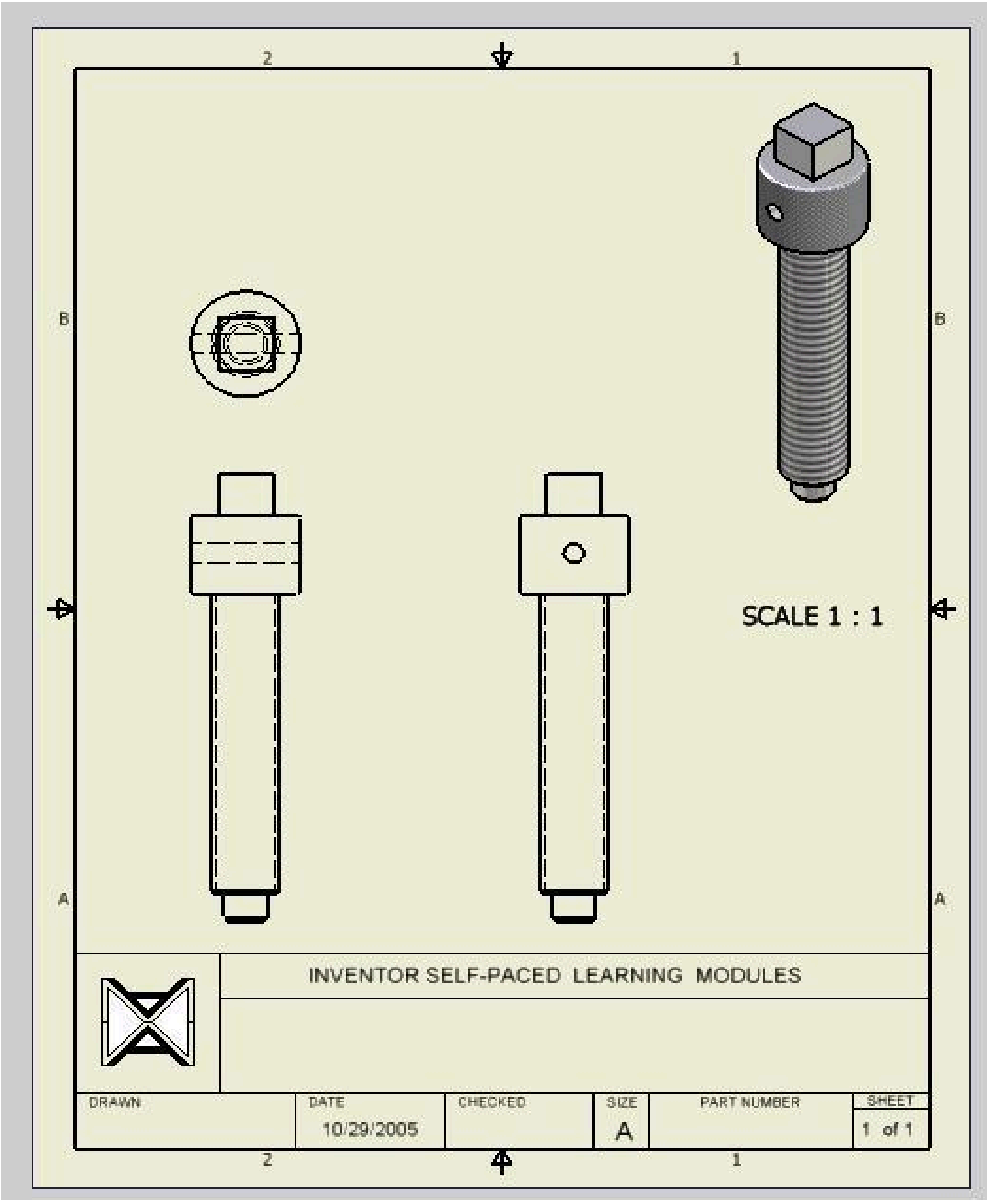


Figure Step 1C [Click to see image full size]

Assembly: Tool Holder
Drawing Size: A
Drawing Name: Inventor Lab 24-1.idw
Part Name: Inventor Workalong 22-1.iam
Template: English-Modules Drawing ANSI (in).idw (Figure Step 1D)

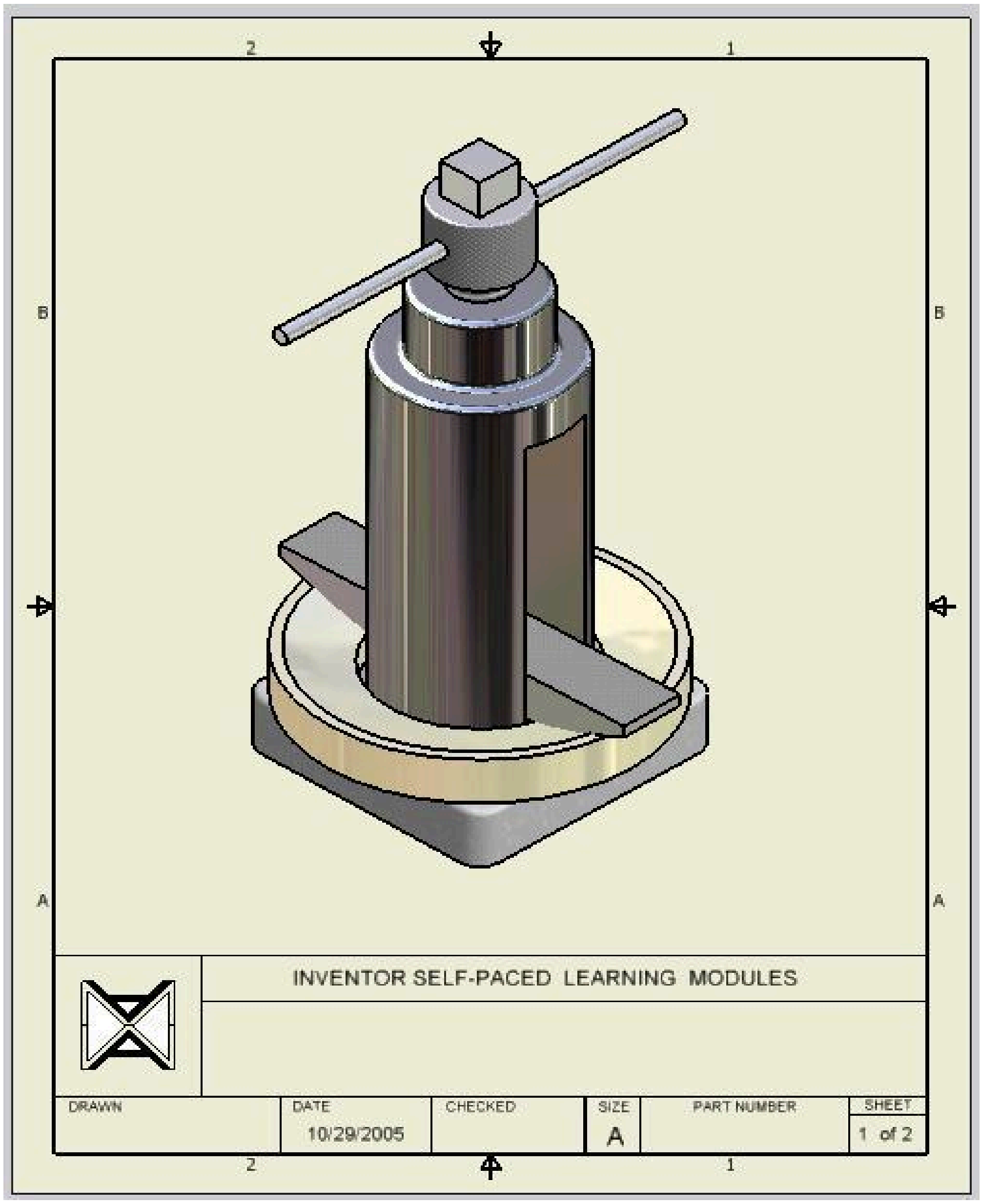


Figure Step 1D [Click to see image full size]

Module 25 2D Drawings – Part 2

Learning Outcomes

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

1. Describe basic dimensioning, centerlines, and standard styles.
2. Describe and apply the STYLES EDITOR command to copy and edit standard styles to create your own dimension and text styles.
3. Describe and apply the RETRIEVE DIMENSION, CENTERLINES, and GENERAL DIMENSION commands to place model and drawing dimensions and centerlines on drawing views.
4. Describe how styles are exported out of and imported into drawing files.

Drafting Lesson: Basic Dimensioning Terms

Figure 25-1 shows the basic dimensioning terms that you will need to know when setting the standards for dimensioning style.

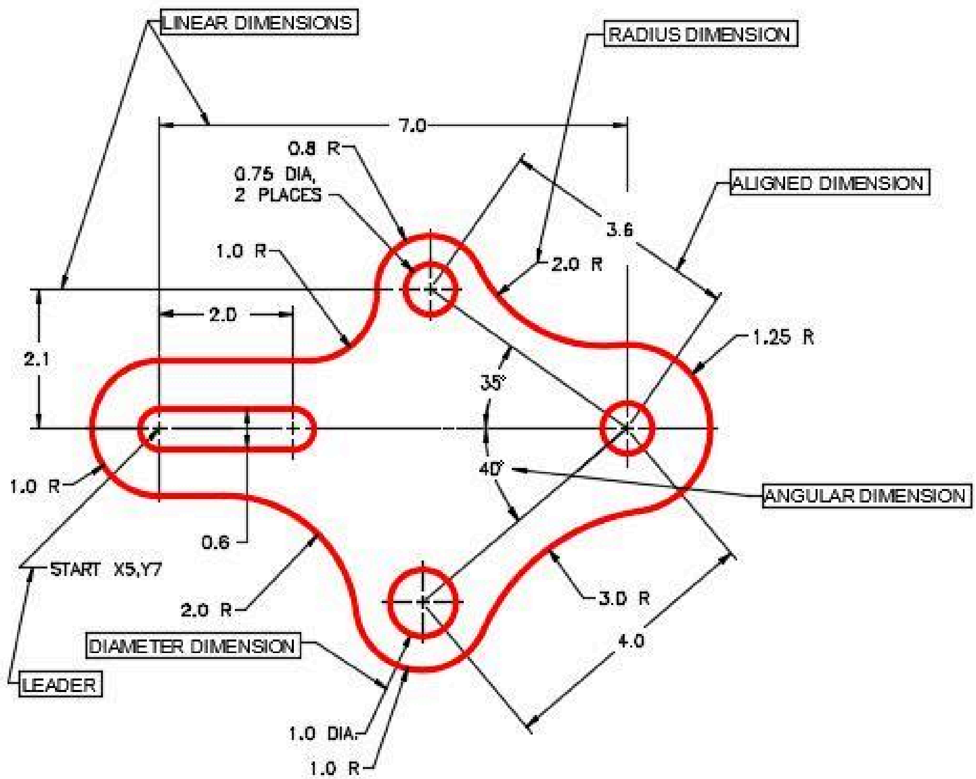


Figure 25-1
Basic Dimensioning Terminology [Click to see image full size]

Dimensioning

Dimensioning is the process of adding size descriptions to the orthographic views of the model that are placed on a drawing. Once a orthographic views of the model are dimensioned, the drawing sheet can then be plotted and used for construction or reference. Up to this point in the modules only shape and size descriptions have been added to the solid models that are been constructed by adding geometrical and dimensional constraints. Since Inventor knows the exact size of the solid models, the drafter/designer only has to control which dimensions are shown and where to position them.
Dimensioning is a complex subject, not difficult, but there is a lot to learn. Therefore, in the Inventor Modules, learning to control the appearance and location of the basic dimension types is taught. As the Inventor user gets more experience dimensioning he/she can experiment with some of the advanced dimensioning features.

Dimensioning Styles

A *dimensioning style* is a named set of variables or settings that controls the way the dimensions will appear on the drawing. There are many different settings in a dimensioning style so it will take the user some time to get used to setting them. Be patient and practice editing styles and inserting dimensions as often as possible.
Inventor comes with several preset dimensioning styles which are part of the template that was used when the drawing file was created. They can be edited but cannot be renamed. It is better to create a new style by copying an renaming one of the Inventor standards and make the necessary changes to it.

After the editing is complete, name the style with an appropriate name and save it so that it can be identified and used again at a later date. Dimensioning styles can also be exported and imported from one drawing to another.

Text Styles

A *text style* is a named set of variables or settings that controls the way text will appear on a drawing. Inventor comes with several preset text styles which were part of the template that was used when the drawing file was created. Although they can be edited, they cannot be renamed. If changes are required, it is better to create a new text style by copying one of the Inventor standards and make the necessary changes to it. After the editing is complete name the style with an appropriate name and save it so that it can identified and used at a later date. Text styles can also be exported and imported from one drawing to another.

Centerlines

A *centerline* is used in technical drawings to indicate the location of the axis of symmetry. Placing centerlines on all objects that have a symmetrical shape will help others who are reading the drawing. The proper use of centerlines also cut down on the number of dimensions that are required on a drawing to fully describe the object.

Placing Dimensions

The two types dimensions that can placed on a drawing are model dimensions and drawing dimensions.

Model Dimensions

Model dimensions are the driven and driving dimensions that were placed in the 2D sketch when the model was being constructed. The RETRIEVE DIMENSION command is used to retrieve the dimensions from the model and display them on the drawing. If the driven dimensions in the original sketches are changed in the future, the model dimensions in the drawing will automatically change since they are showing the actual size of the model.

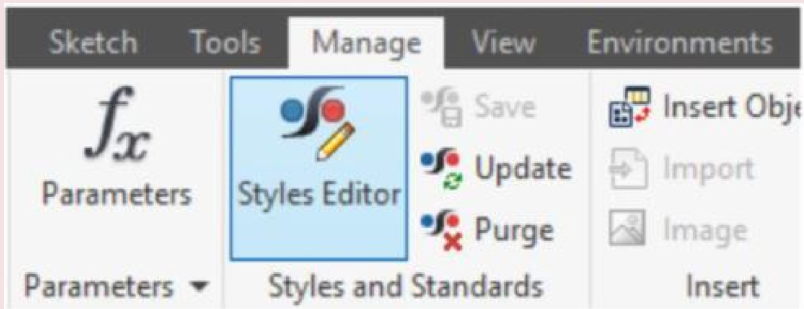
Drawing Dimensions

Drawing dimensions are dimensions that are placed in the drawing using the GENERAL DIMENSION command. Dimensions are placed exactly where they are located in the 2D sketch when the model was created. Inventor will obtain the actual sizes and if the model is changed in the future, the drawing dimensions will automatically change to reflect the new size of the model.

Inventor Command: STYLES EDITOR

The STYLES EDITOR command is used to create and/or edit the styles and standards used by a drawing file.

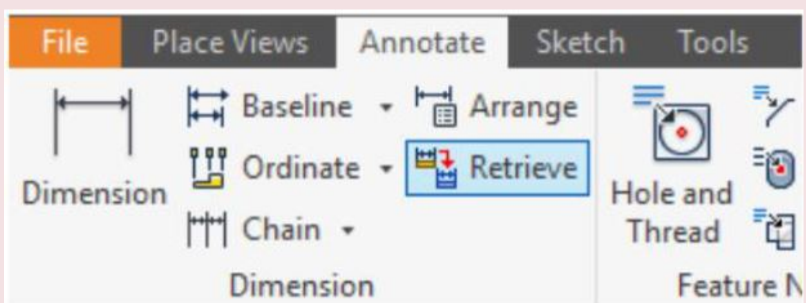
Shortcut: **none**



Inventor Command: RETRIEVE DIMENSIONS

The RETRIEVE DIMENSIONS command is used to retrieve model dimensions from the sketches used to create the solid model. Only dimensions that are parallel to the plane of the view will display. The user can select the dimensions that they want to display on the drawing.

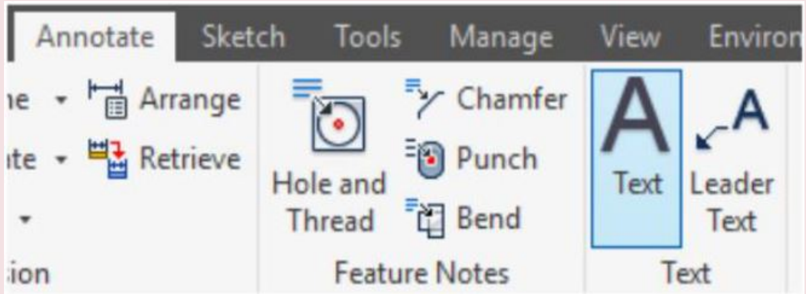
Shortcut: **none**



Inventor Command: TEXT

The TEXT command is used to place text on the a drawing sheet. It uses the default text style when the text is inserted.

Shortcut: **none**



Drafting Lesson: Center Lines

In multiview drawings, the *center line* is used to indicate the location of the axis of symmetry. Placing centre lines on all objects that have a symmetrical shape will help the reader and save the you from inserting a lot of dimensions as you will see in future modules. Below are some examples of typical applications of the use of centre lines in a multiview drawing.

A ' C ' with an ' L ' through it is the symbol for a centre line. Center lines are drawn as repeating long and short lines. See Figure 37-2.

Figure 37-2

Figure 37-3 shows a centre line indicating the centre of the circle with two short lines called a *center mark* intersecting at the centre. Note in the right side view, the centre line follows the length of the cylinder.

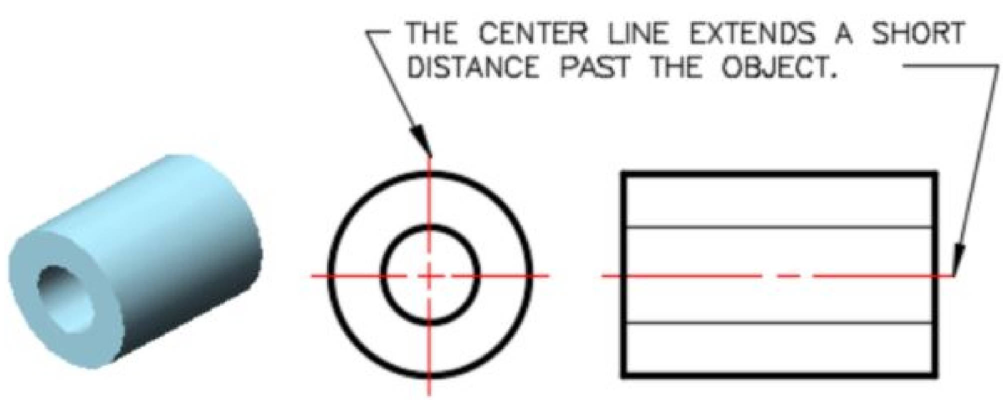


Figure Step 37-3

When the symmetry ends, so does the centre line. Note how the centre line ends on the left side of the arc. See Figure 37-4

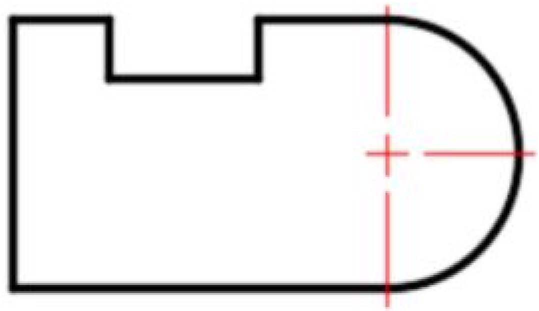


Figure Step 37-4

Figure 37-5 shows the centre lines along with the hidden lines that indicate a hole going through the object. The centre line on the top and bottom circles stops at the circle.

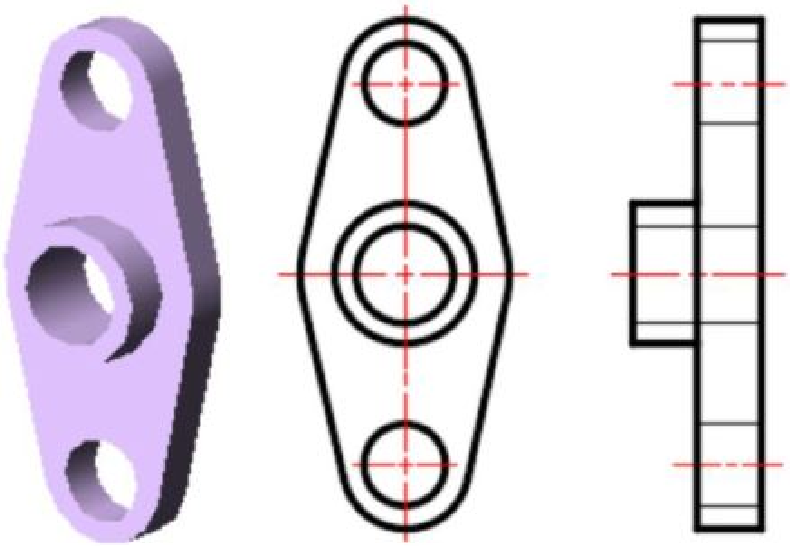


Figure Step 37-5

In Figure 37-6, note how centre lines are drawn for an array of circles.

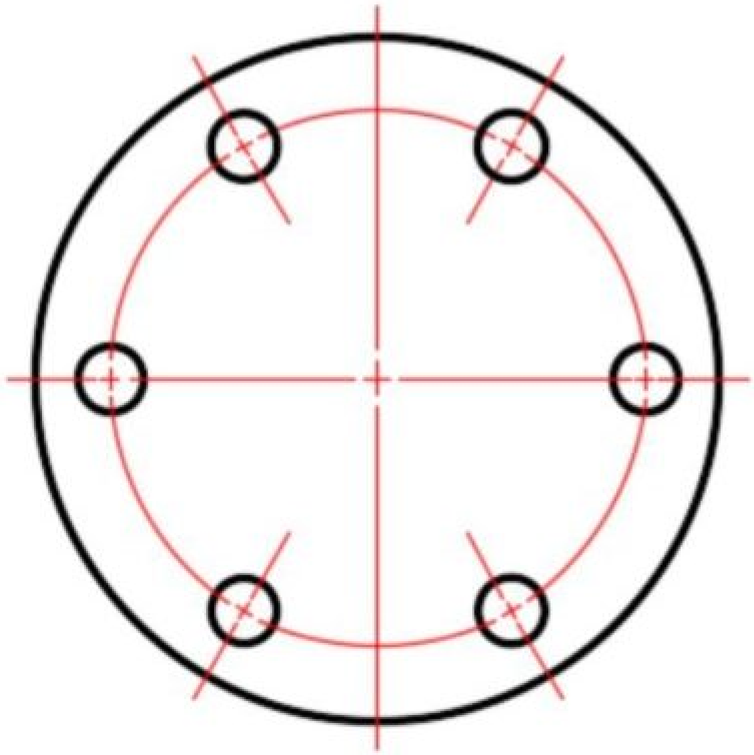


Figure Step 37-6

There are four icons used to place centerlines on the drawing. In this module, you will be using three of them. The Center Mark, Centerline, and Centered Pattern icons are shown in Diagrams F1, G1 and H1. Diagrams F2, G2 and H2 show what centerline the applicable icon will place.

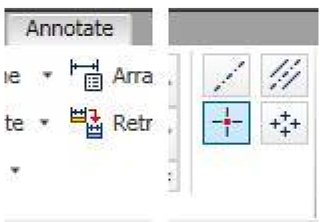


DIAGRAM F1

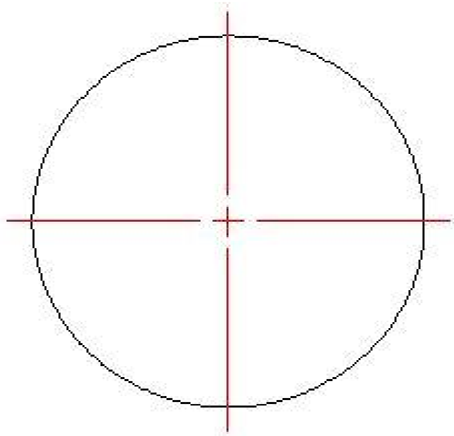


DIAGRAM F2

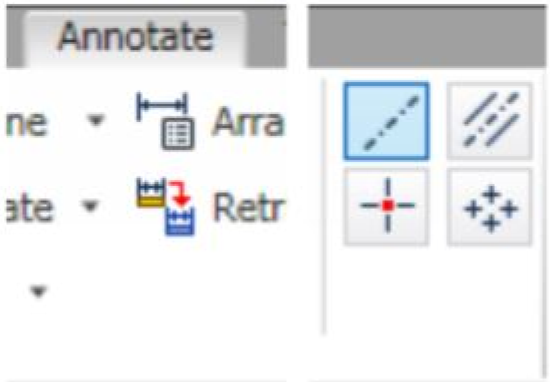


DIAGRAM G1

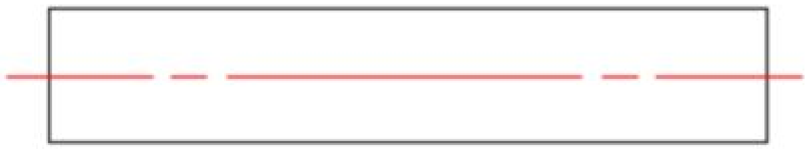


DIAGRAM G2

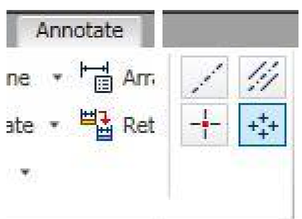


DIAGRAM H1

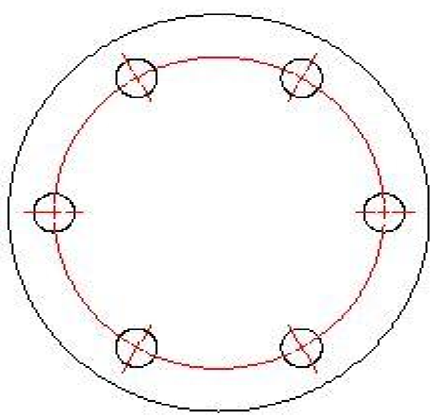


DIAGRAM H2

WORK ALONG: Creating Dimensioning and Text Styles

Step 1

Check the default project and if necessary, set it to: Inventor Course.

Step 2

Open the drawing file: Inventor Lab 24-1A.idw. (Figure Step 2)

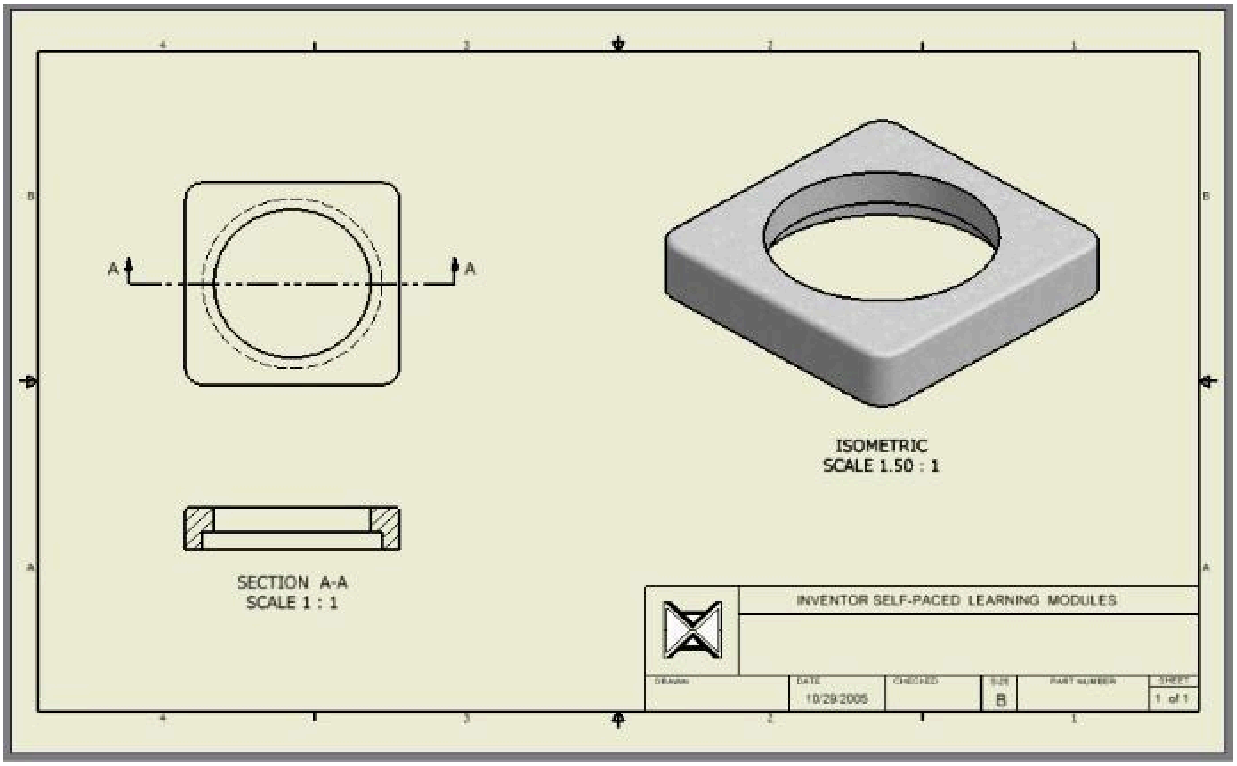


Figure Step 2 [Click to see image full size]

Step 3

Enter the STYLES EDITOR command to open the Styles and Standard Editor dialogue box. Expand the children under the heading Dimension from the list of styles and standards on the left side of the dialogue box. The seven dimensioning styles listed are the styles that are already contained in your drawing. They were in the template file that you used when you created the drawing. (Figure Step 3)

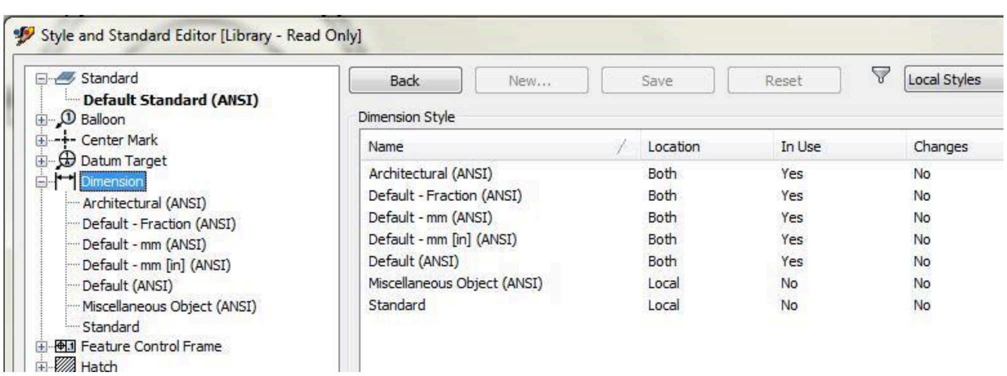


Figure Step 3

Step 4

Right-click the dimensioning style: Default (ANSI). In the New Style Name dialogue box, enter the name: Modules in ANSI. Ensure that you enable the Add to Standard and then click OK. (Figure Step 4A and 4B)

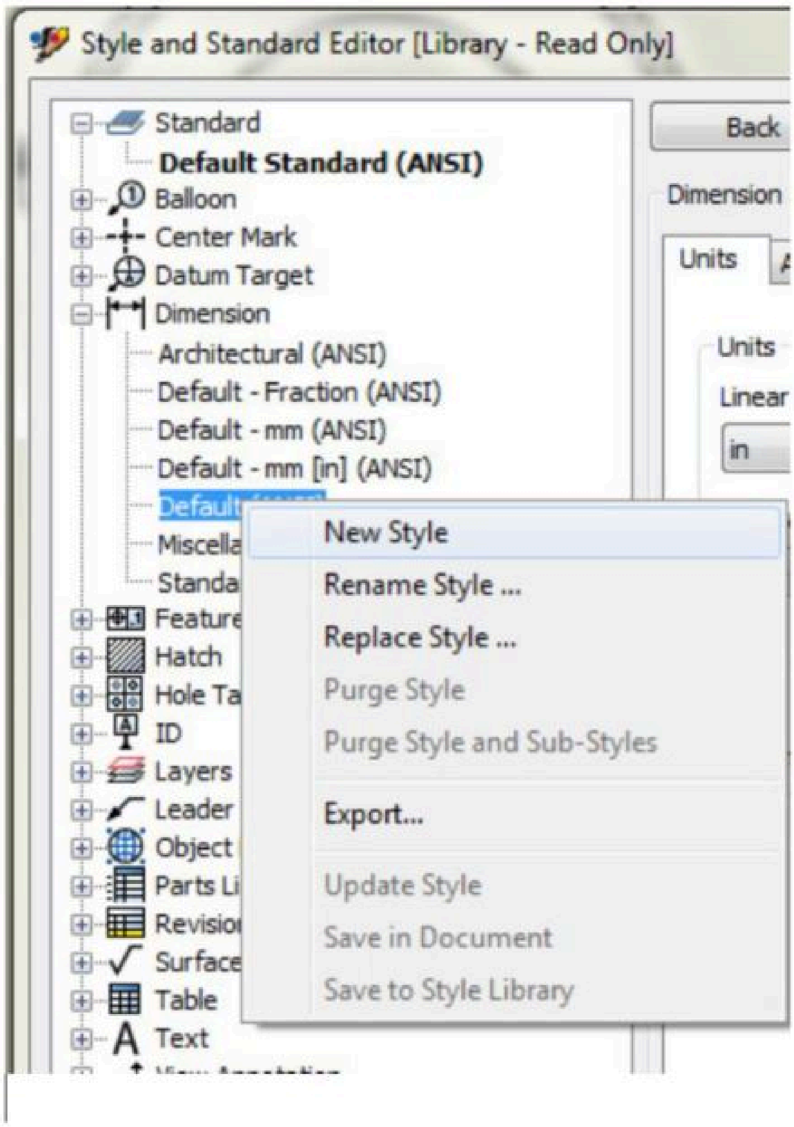


Figure Step 4A

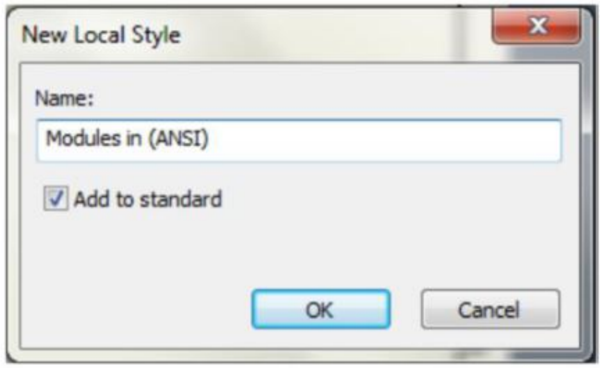


Figure Step 4B

AUTHOR'S COMMENTS: You are copying the Default ANSI (in) dimensioning style to create your own style. It is always best to copy from an existing style. That way, you get the existing standards and you only have to change the items that you require to suit your new style.

Step 5

Click the dimensioning style: Modules in (ANSI) in the style list on the left side of the dialogue box to make it the current style. Note that on the right side of the dialogue box your newly created style is listed at the top above the tabs. This indicates it can be edited in the dialogue box. Enable the Units tab. (Figure Step 5A and 5B).

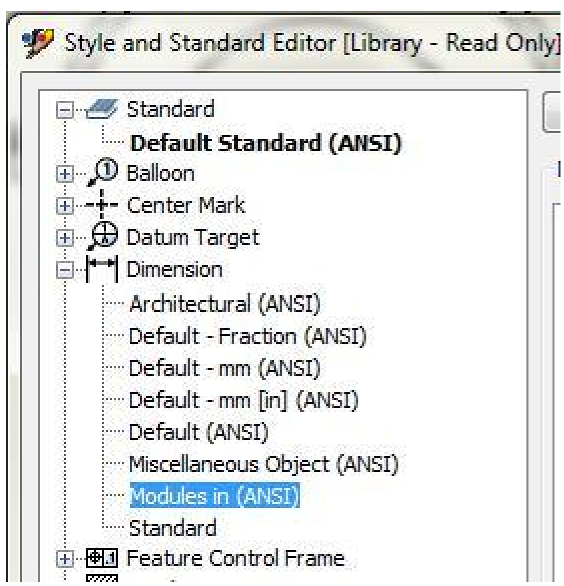


Figure Step 5A

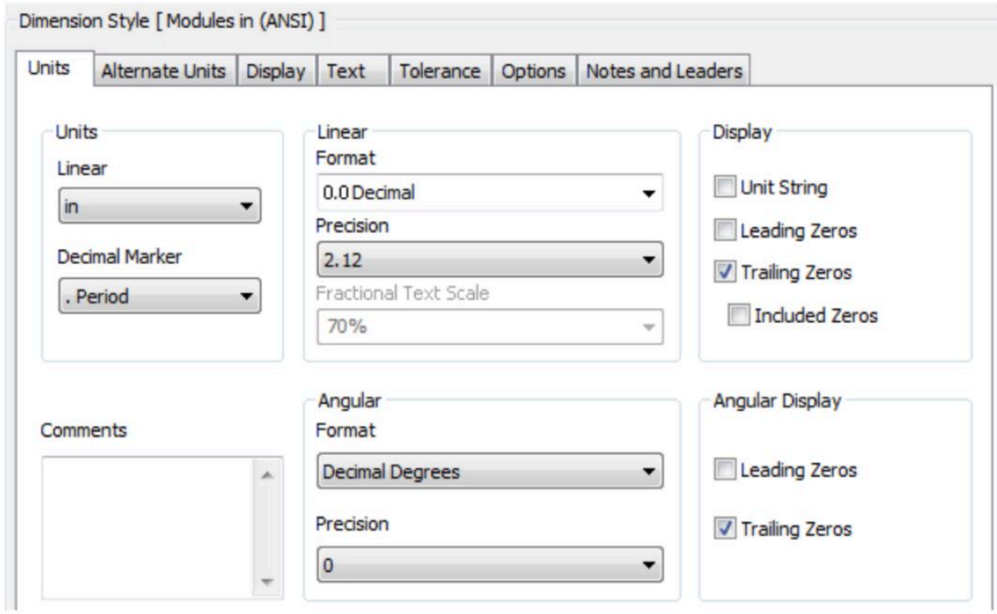


Figure Step 5B

Step 6

Change the Linear and Angular box to match the settings shown in the figure. (Figure Step 6)

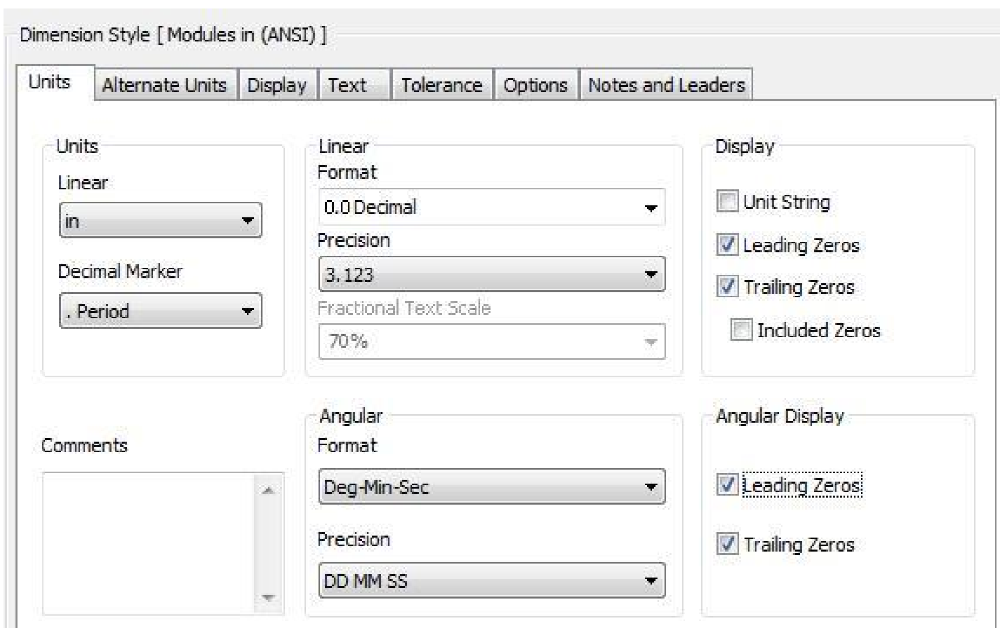


Figure Step 6

Step 7

Enable the Display tab and if necessary, make any changes until it matches the figure. (Figure Step 7)

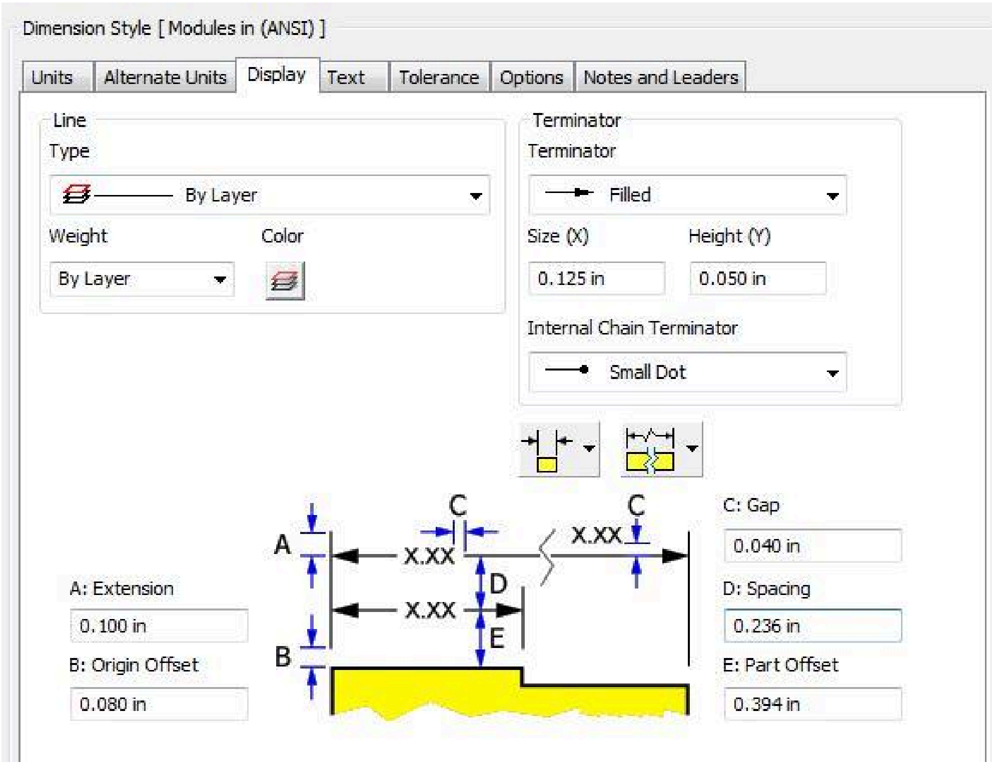


Figure Step 7

Step 8

Enable the Options tab and if necessary, make any changes until it matches the figure. (Figure Step 8)

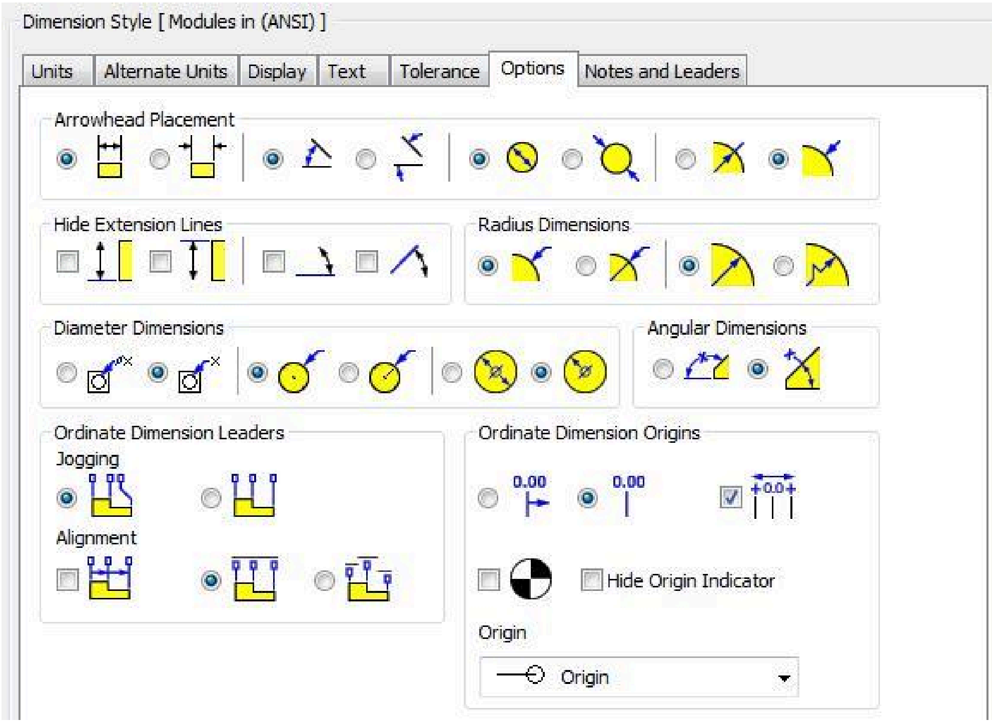


Figure Step 8

Step 9

Enable the Notes and Leaders tab and if necessary, make any changes until it matches the figure. (Figure Step 9)

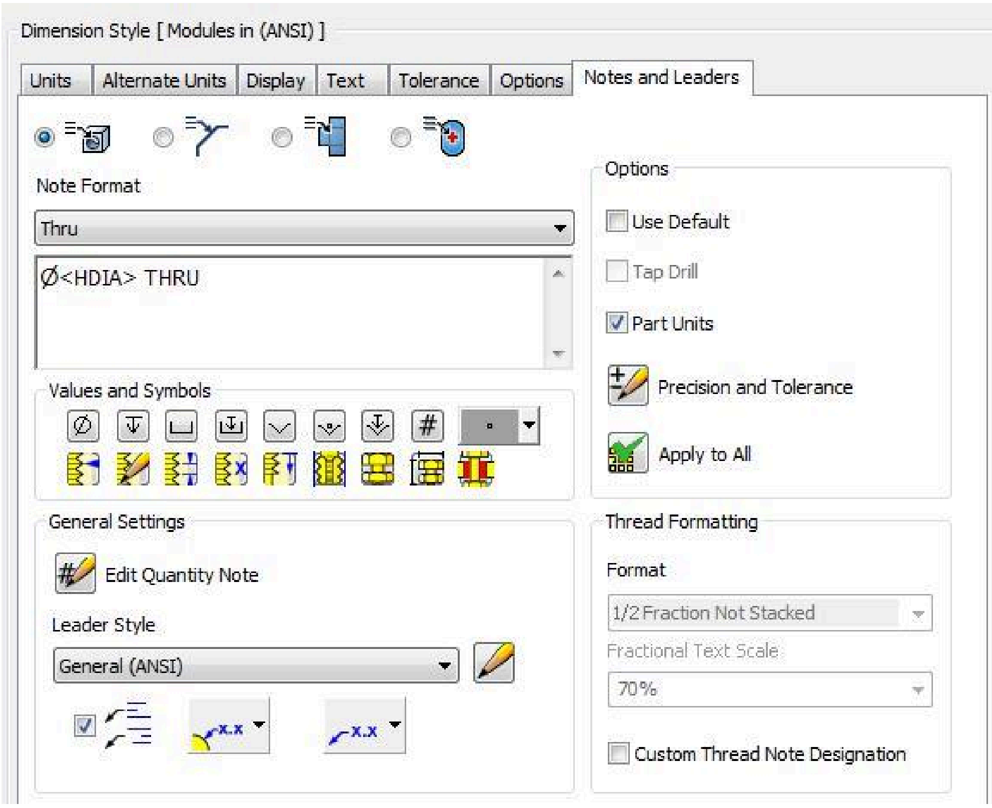


Figure Step 9

Step 10

Click the Save button to save the changes you made to the dimensioning style: <u>Modules in ANSI</u>. Click <u>Done</u> to close the dialogue box. (Figure Step 10)

Figure Step 10

MUST KNOW: The default dimension styles cannot be renamed nor should they be altered. It is always best to create your own style using a default style to copy from. After you edit it to make the changes to suit the required style, name and save the style. Pick an appropriate name so that you can easily find and use it in the future.

Step 11

Expand the children under the heading Text from the list of styles and standards on the left side of the dialogue box. The style names listed are the text styles that are already contained in your drawing. (Figure Step 11)

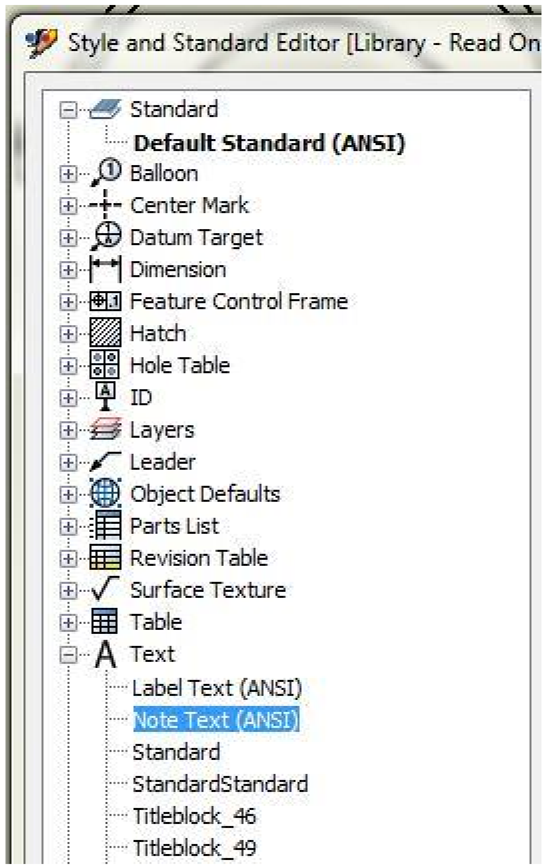


Figure Step 11

Step 12

Right click the style: Note Text (ANSI). In the right-click menu, select New Style. (Figure Step 12)

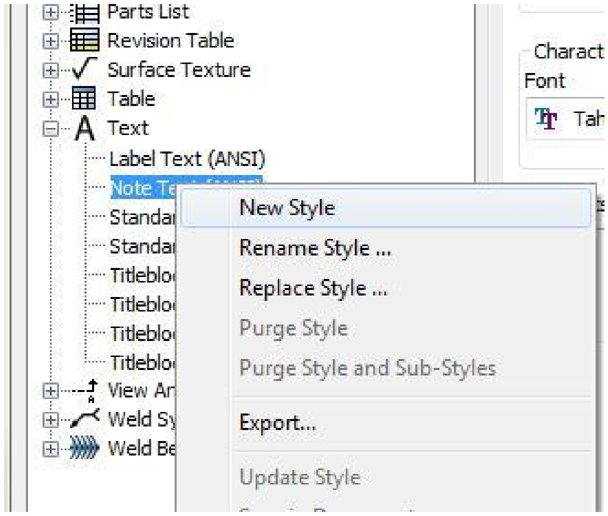


Figure Step 12

Step 13

This will open the New Style Name dialogue box as shown in the figure. Note that since the text style: Note Text (ANSI) was the current style, you will be starting with a copy of its settings. Enter the name: Modules Note Text (ANSI) and click OK. Ensure that Add to standard is enabled. (Figure Step 13)

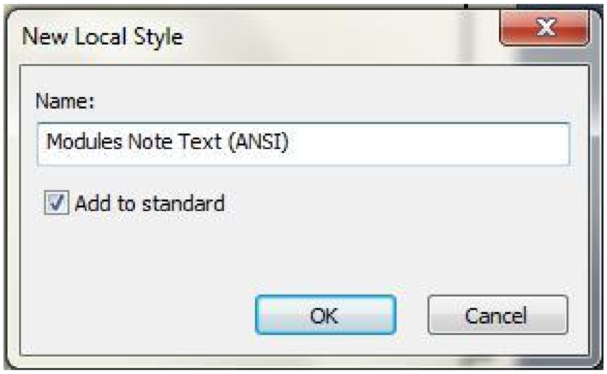


Figure Step 13

Step 14

Make any changes necessary to match the figure. (Figure Step 14)

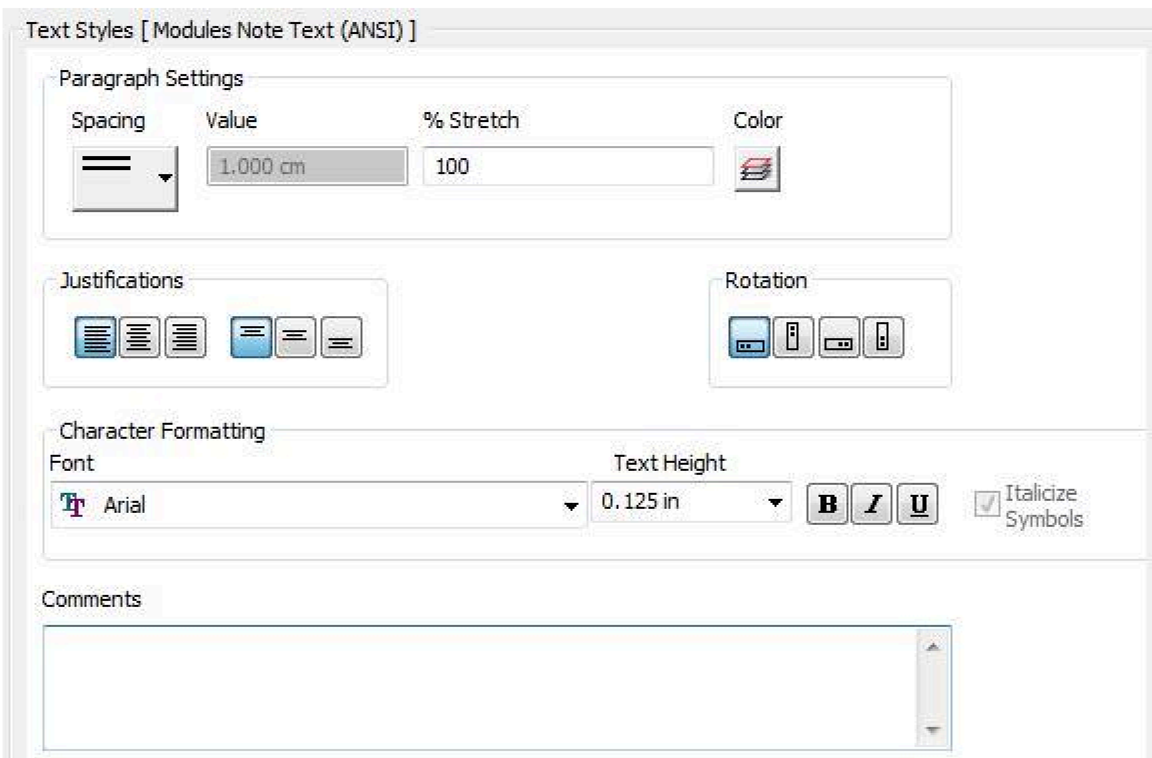


Figure Step 14

Step 15

Click the Save button to save the changes you made to the text style.

Step 16

Use what you learned earlier in the workalong, make the dimensioning style: Modules in ANSI the active style. (Figure Step 16)

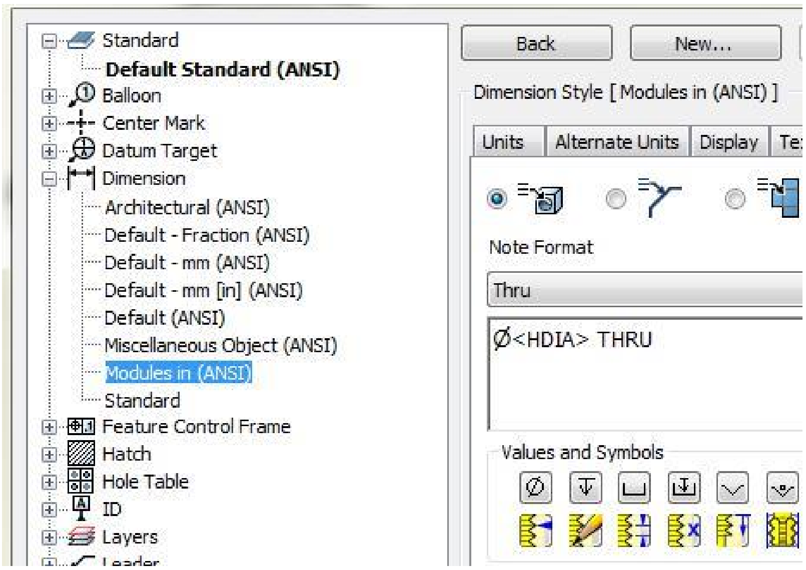


Figure Step 16

Step 17

Enable the Text tab. Pull down the Primary Text Style pull-down box and select the text style: Modules Note Text (ANS)I. This will make it the default text style for the dimensioning style: Modules in ANSI. (Figure Step 17)

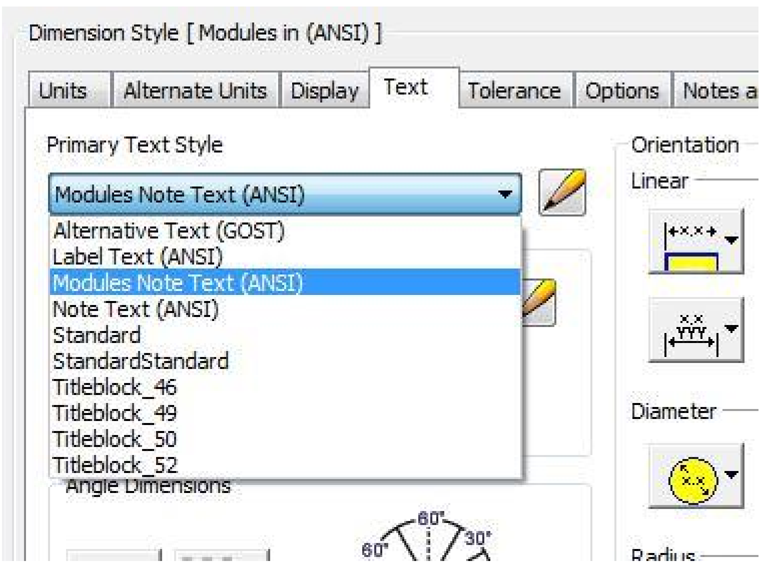


Figure Step 17

Step 18

The Text tab in the active dimension style: Modules in ANSI should appear as shown in the figure.

Step 19

Click the Done button and if you are asked to Save Edits, click Yes. (Figure Step 19)

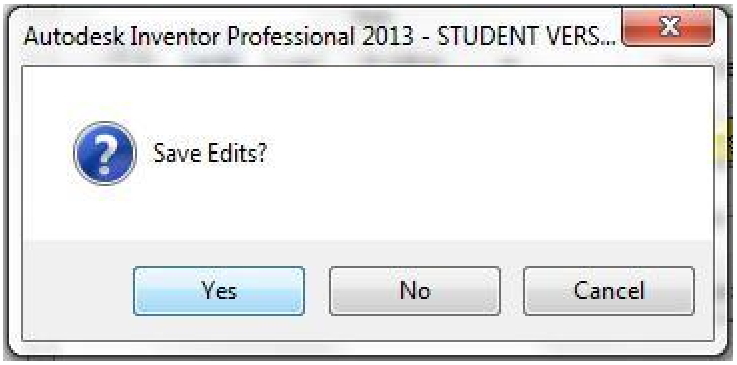


Figure Step 19

Step 20

Save and close the drawing file.

WORK ALONG: Adding Annotation to the Drawing

Step 1

Check the default project and if necessary, set it to: Inventor Course.

Step 2

Open the drawing file: Inventor Lab 24-1A.idw.

Step 3

Click the Centerline icon to place the vertical centerline on the Top view. For the first point, snap to the midpoint of the line on the top of the view. For the second point, snap to the midpoint of the bottom line of the Top view. Move the cursor a short distance below the Top view and click it to indicate the distance the centerline is to go past the view. (Figure Step 3A, 3B, 3C, and 3D)

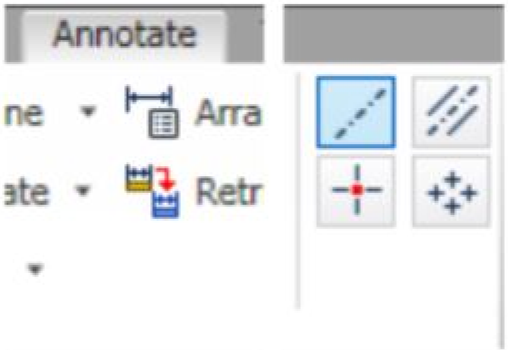


Figure Step 3A

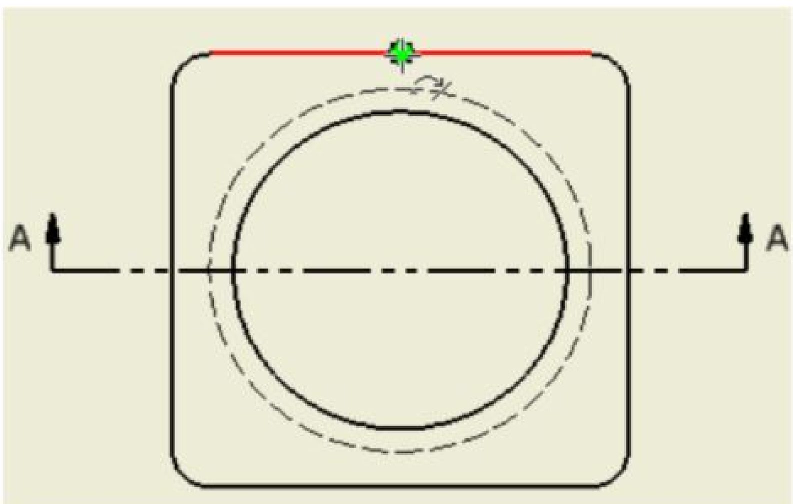


Figure Step 3B

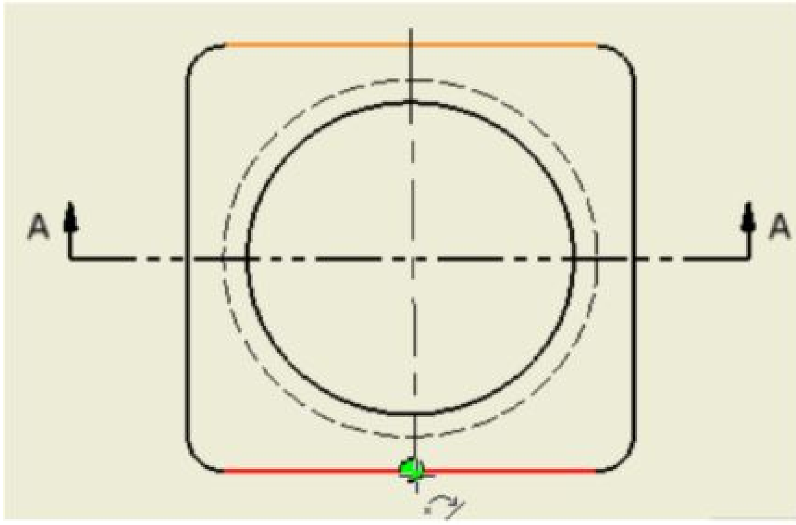


Figure Step 3C

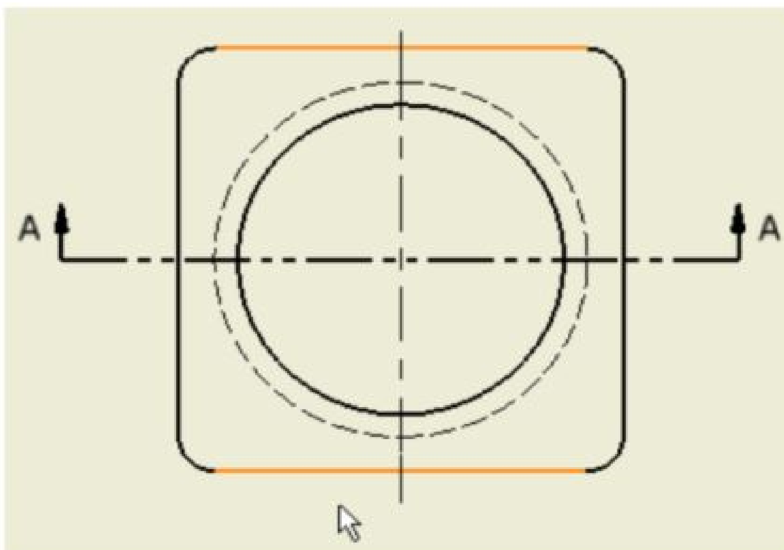


Figure Step 3D

Step 4

Using what you just learned, place the centerline on the Front view. (Figure Step 4)

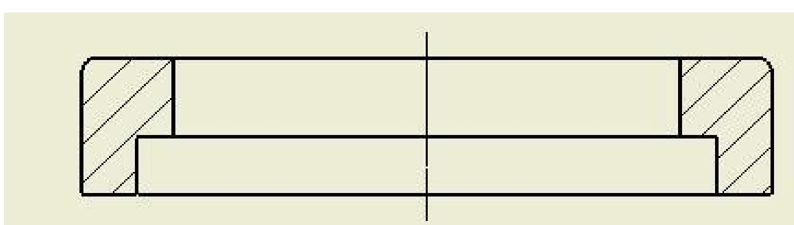


Figure Step 4

AUTHOR'S COMMENTS: Note that because the centerline in the Front view is so short, the break lines do not appear.

Step 5

Open the Style and Standard Editor dialogue box. On the left side, expand the children in the Center Mark heading. Select Center Mark (ANSI) to make it the current style. (Figure Step 5)

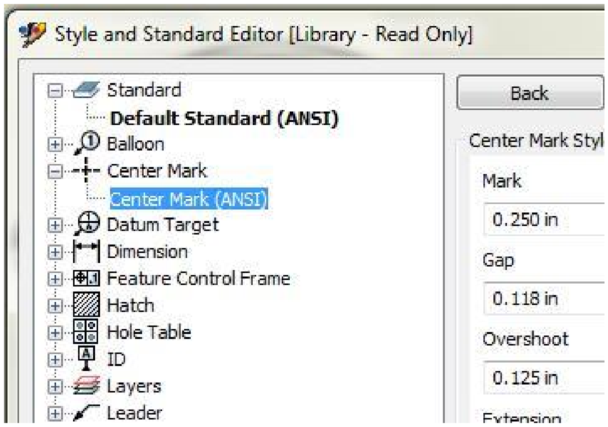


Figure Step 5

Step 6

Right-click the standard style: Center Mark (ANSI) and click New Style. In the New Style Name dialogue box, enter the name: Modules Center Line Short (ANSI). (Figure Step 6)

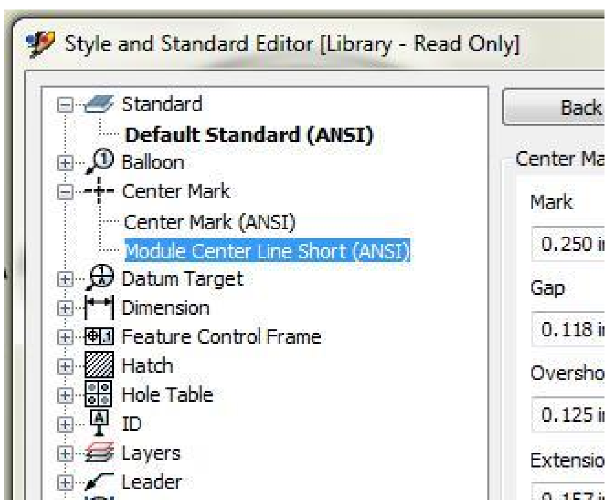


Figure Step 6

Step 7

Select the new style that you just created to make it the current style. Make the changes shown in the figure. (Figure Step 7)

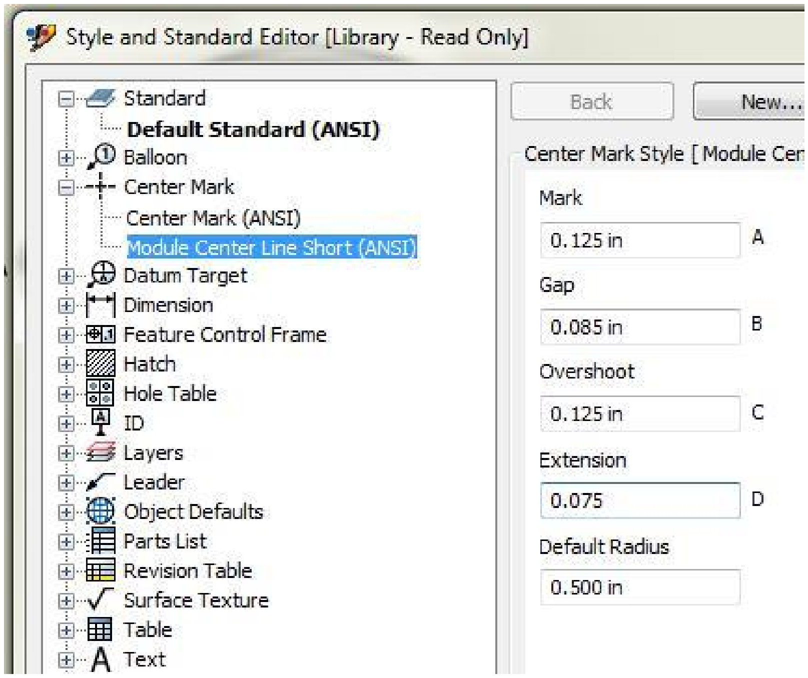


Figure Step 7

Step 8

Select the centerline in the Front view. It will highlight. (Figure Step 8)

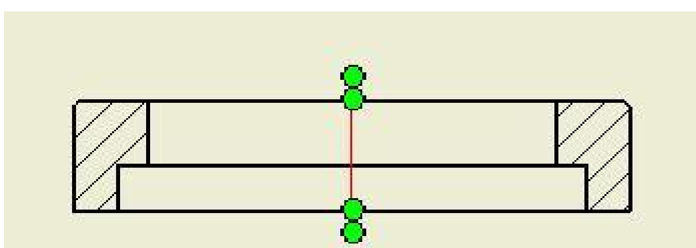


Figure Step 8

Step 9

With the centerline selected, look on the Inventor Standard pull-down menu. Note how it shows Standard as the centerline type for selected object. Select the centerline style: Modules Center Line Short ANSI. (Figure Step 9A and 9B)

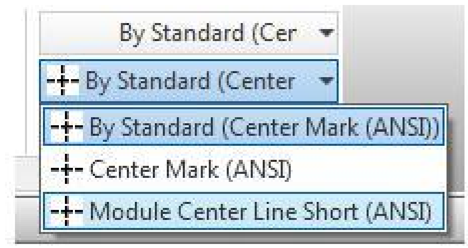


Figure Step 9A

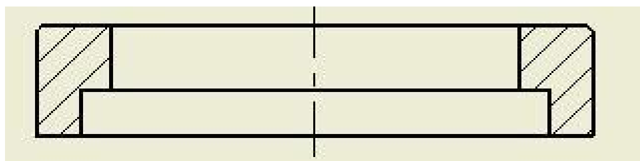


Figure Step 9B

USER TIP: When creating and dimensioning the 2D sketches in future models, keep in mind how the RETRIEVE DIMENSION command works. It retrieves both driven and driving dimensions that are parallel to the plane of each vew. After placing the driven dimension to constrain the sketch, add the necessary driving

dimensions so that when you create the drawing and dimension the model, it will retrieve all of the dimensions required. That way, you will may not have to place any general dimensions.

Step 10

Enter the RETRIEVE DIMENSION command and in the Retrieve Dimensions dialogue box, ensure that Select View icon is enabled. Select the Top view. (Figure Step 10)

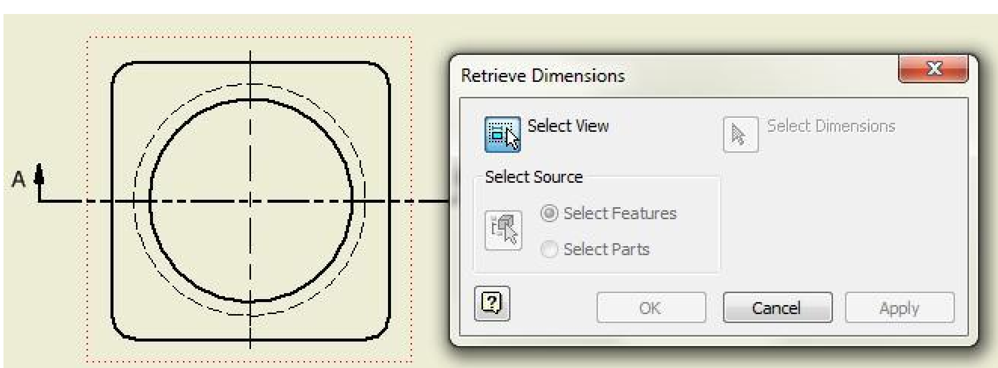


Figure Step 10 [Click to see image full size]

Step 11

In the Select Source box, enable Select Parts. Select all objects. The dimensions that you inserted in the sketches that are parallel to the plane will display. (Figure Step 11)

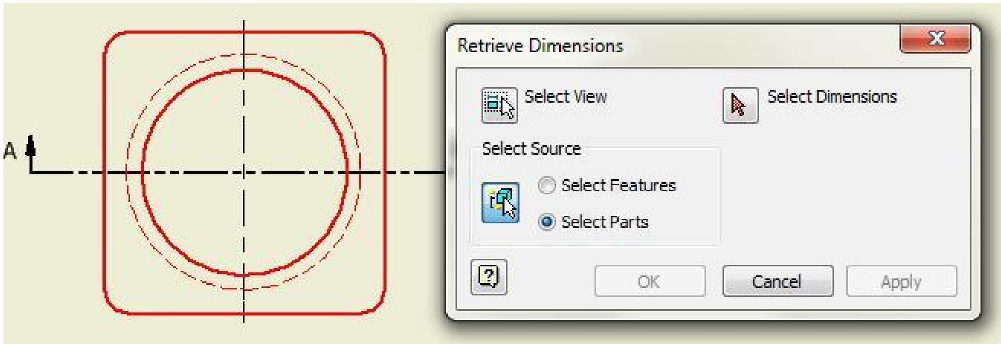


Figure Step 11 [Click to see image full size]

Step 12

Ensure that the Select Dimension icon is enabled select the dimension(s) that are selected in the figure. (Figure Step 12)

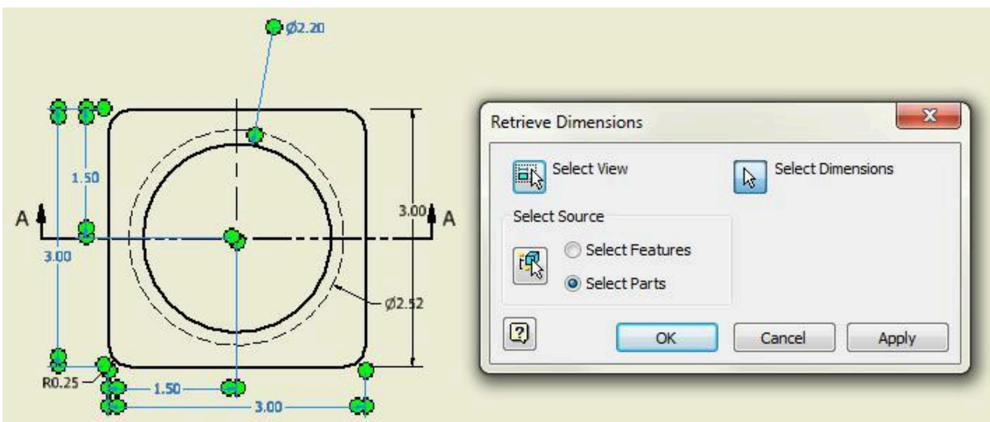


Figure Step 12 [Click to see image full size]

AUTHOR'S COMMENTS: The dimensions that display on the view may be different then the ones shown here. It depends how you constructed the model for this part. Select the dimensions to the best of your ability. If you cannot display all the dimensions shown, you can add them as drawing dimensions later in the workalong.

Step 13

Using what you just learned, select the dimensions for the Front view of the model. In this case, there was only one. (Figure Step 13)

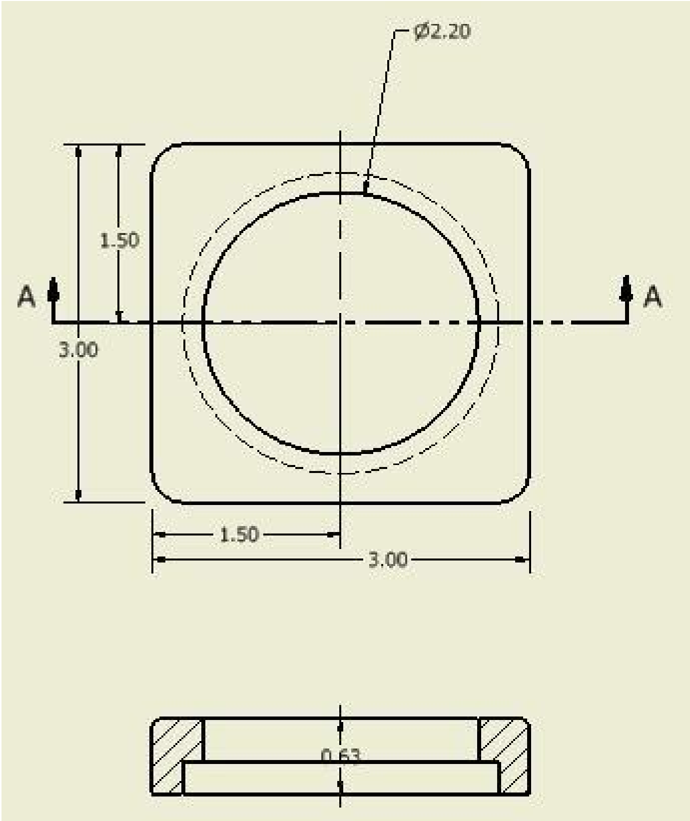
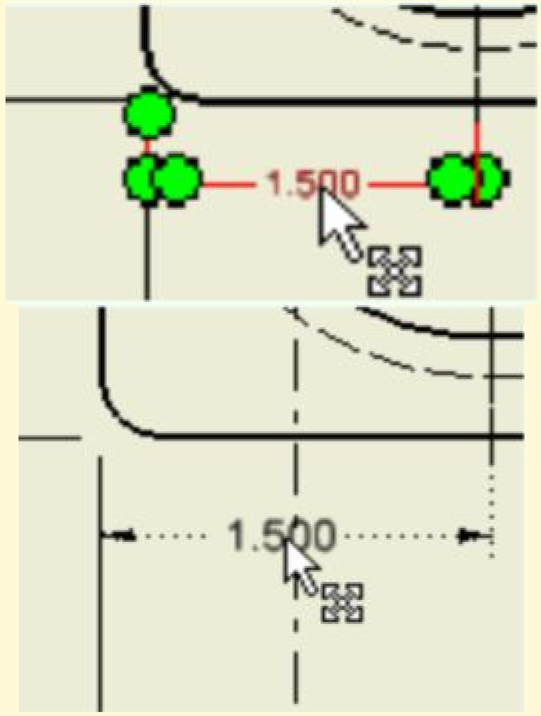


Figure Step 13

AUTHOR'S COMMENTS: The dimensions that display on your model may be different then the one shown here. It depends how you constructed the model for this part. Select the dimension to best of your ability.

USER TIP: To relocated a dimension on a drawing, move the cursor onto the dimension. When the cursor is on a dimension, it will highlight and the Move icon will display as shown in the figure to the right. The Move cursor is shown in the figure on the left. When the Move cursor displays, press and hold down on the left mouse button and drag the cursor to approximately the location you want to place the dimension. As you do that, Inventor will display a temporary centerline indicating the centre location between the extension lines as shown in the figure to the right. It will help you centre the dimension text. When you have the dimension positioned, release the mouse button.


1.500
1.500

Step 14

Move the dimensions to match the figure as close as possible. (Figure Step 14)

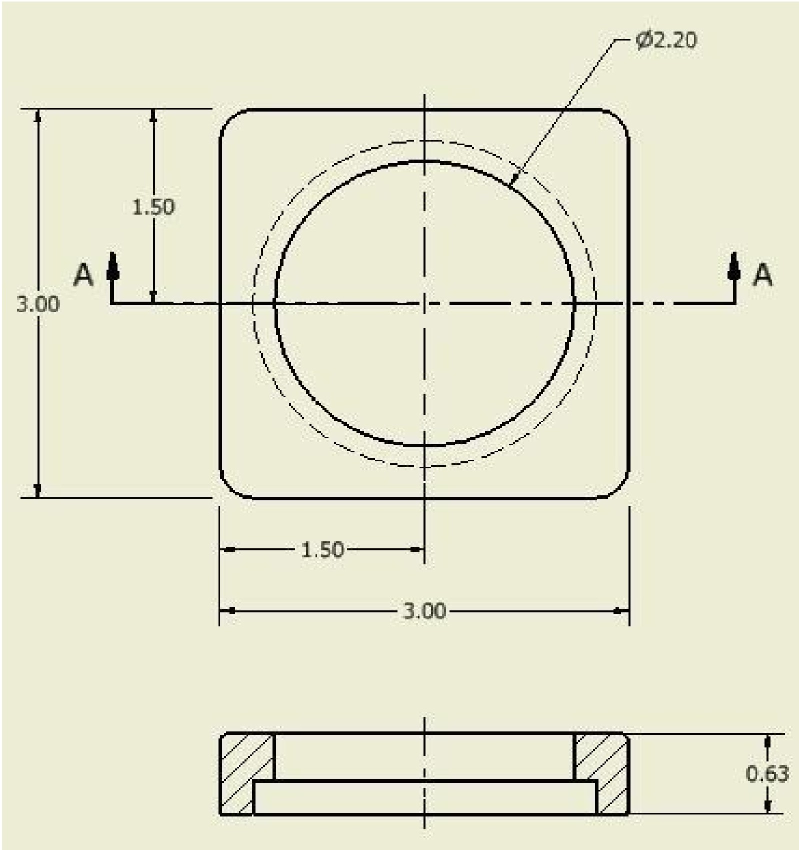


Figure Step 14

AUTHOR'S COMMENTS: Use the move technique shown in the User Tip on Page 25-18. Note: If you are missing some of the dimensions that are shown in the figure, don't be concerned. You can add them as drawing dimensions later in the module.

Step 15

Select all of the dimensions. Change the dimensioning style to: Modules in ANSI. To do this, while the dimensions are selected, pull down the standards list. From the dimension style list, find and select the style: Modules in ANSI in the pull-down list on the Inventor Standard menu. (Figure Step 15A , 15B, and 15C)

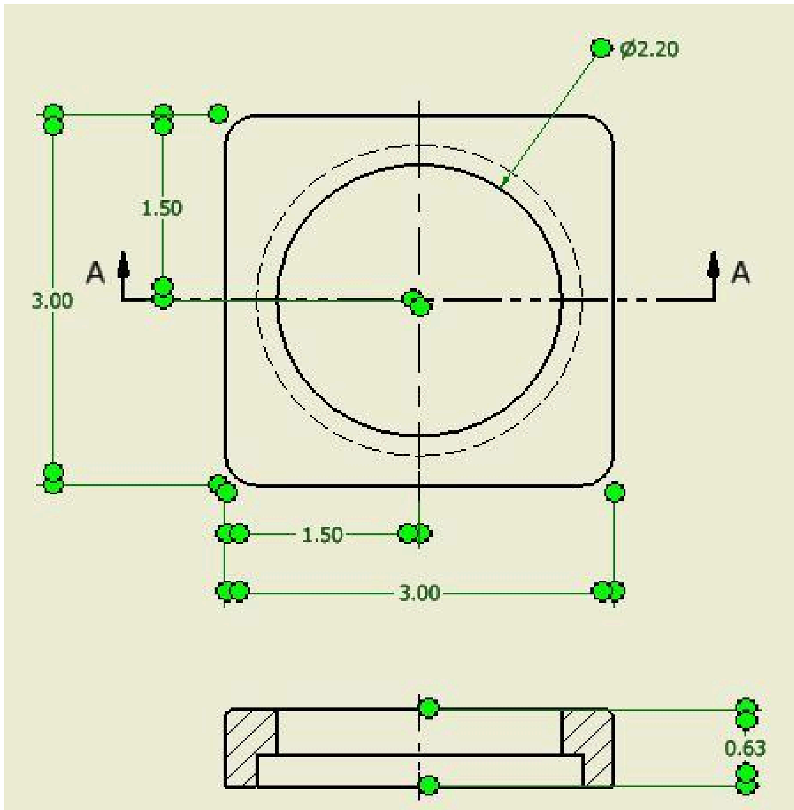


Figure Step 15A [Click to see image full size]

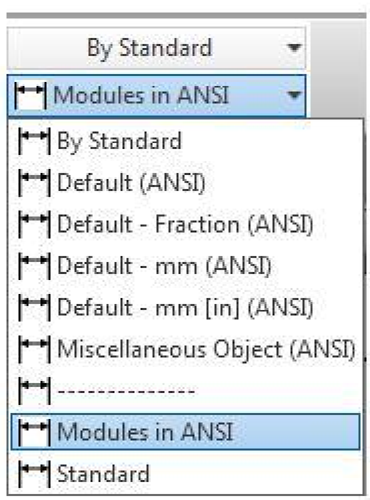


Figure Step 15B

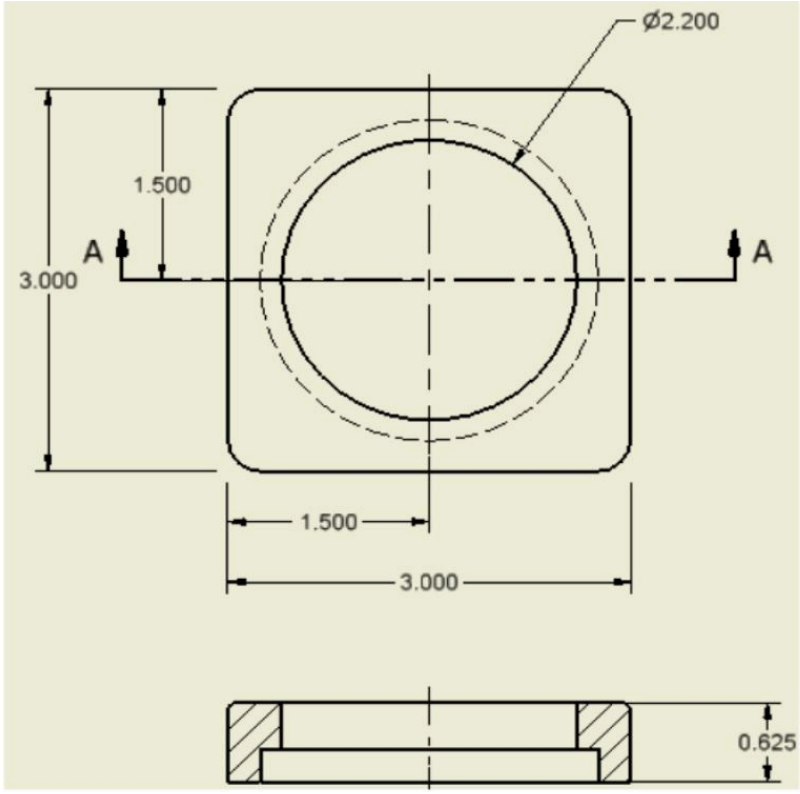


Figure Step 15C [Click to see image full size]

Step 16

Set the default dimensioning style before you insert the drawing dimensions. Pull-down the style standards list and select: Modules in (ANSI). It should now display as shown in the figure. When you insert your dimensions they will use the default dimensioning style. (Figure Step 16)

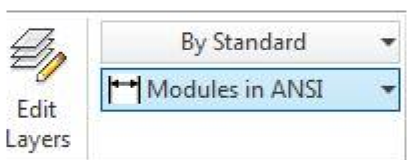


Figure Step 16

Step 17

Add the remainder of the dimensions to match the figure. This is done using the GENERAL DIMENSION (D) command, just like you did when creating your 2D sketches. (Figure Step 17A and17B)

Figure Step 17A

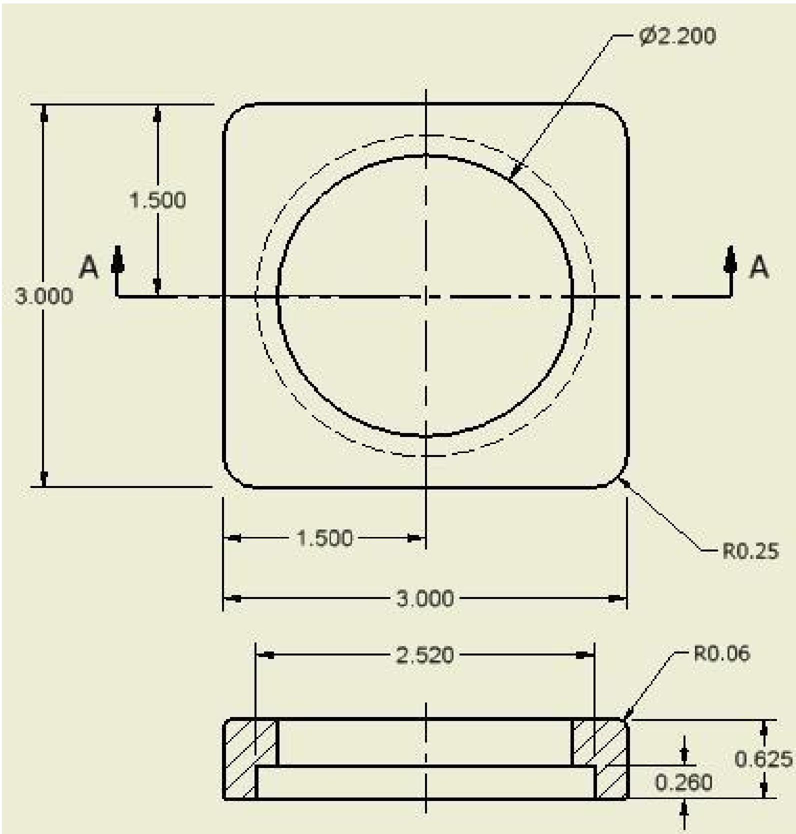


Figure Step 17B [Click to see image full size]

Step 18

To add or edit dimension text, click the text and right-click the mouse. In the Right-click menu, select Text. This will open the Format Text dialogue box. (Figure Step 18)

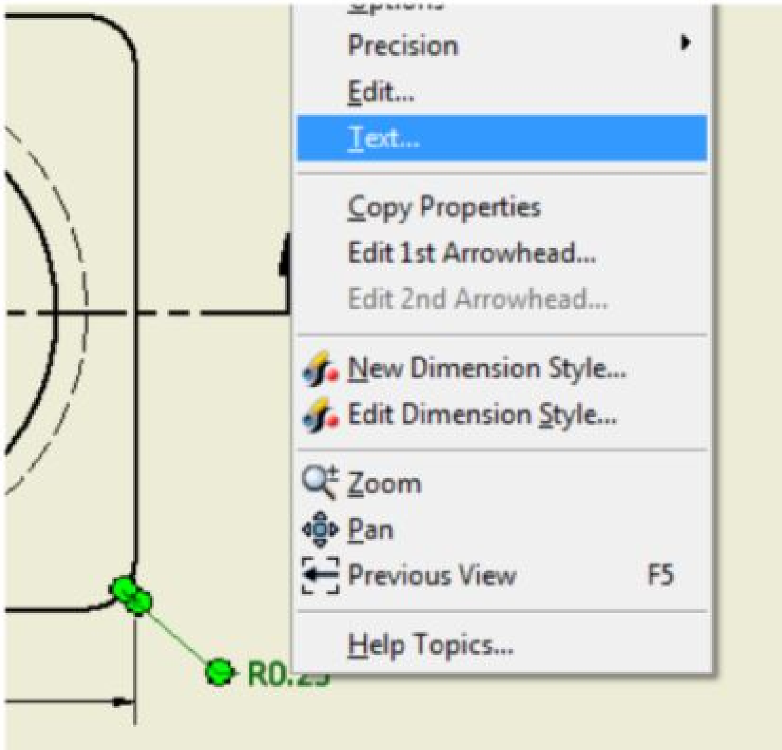


Figure Step 18

Step 19

In the text box at the bottom of the dialogue box you will see the symbols << >>. This symbol indicates the actual model dimension. You cannot edit the dimension since Inventor gets the dimension from the model. To add text, click the cursor behind the symbol and enter a comma. Press the Enter key to go to a new line. Add the Figure Step 18 text 4 PLACES. (Figure Step 19)

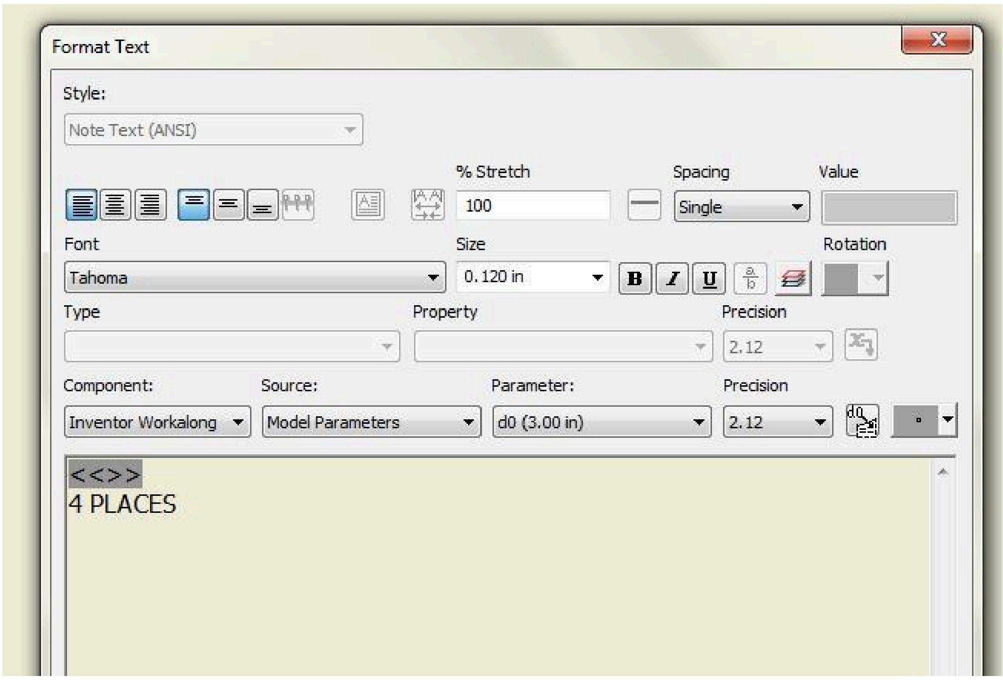


Figure Step 19

MUST KNOW: The dimension text symbol "<< >>" signifies that the actual dimension number is a model or drawing dimension size. Inventor will obtain the actual dimension from the solid model and display it on the drawing. You cannot edit this number nor can you delete it. If the size of the model is modified, the dimension Inventor displays will change to reflect the true size of the model.

Step 20

Using what you just learned, add the text to the other radius dimension. (Figure Step 20)

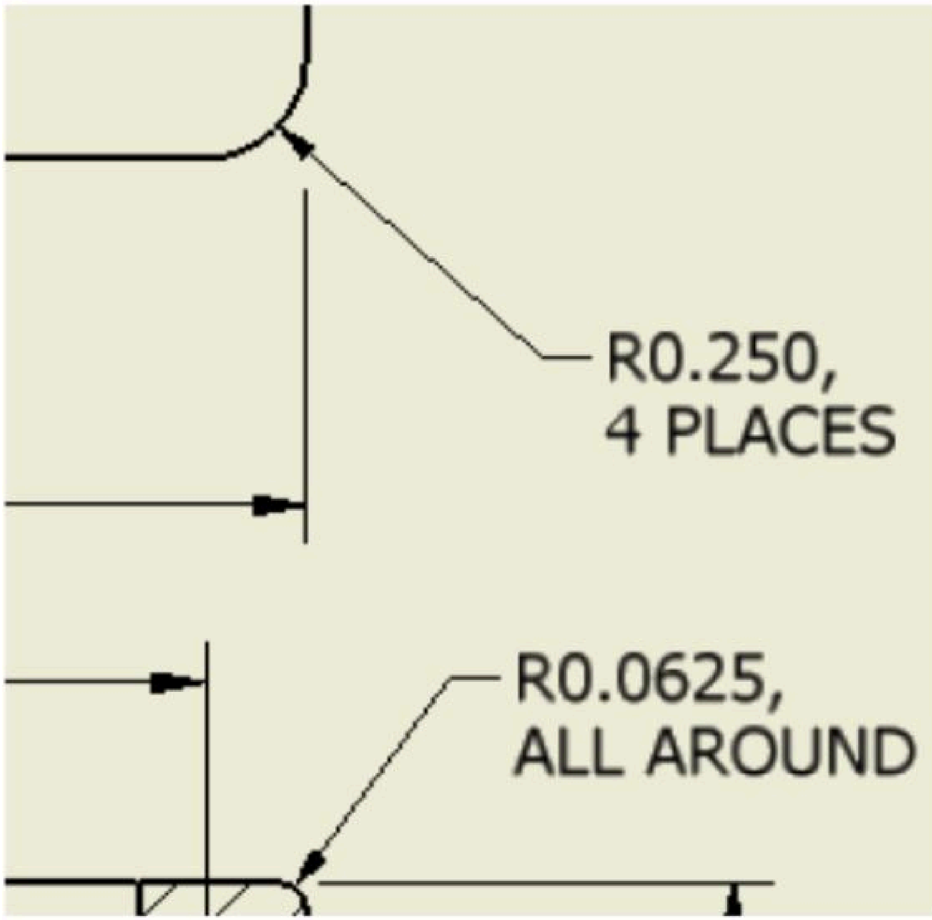


Figure Step 20

Step 21

To fill-in the titleblock, simply change the drawing's properties. To do that, right-click the drawing's icon in the Browser bar. In the Right-click menu, click iProperties. The titleblock has been programmed to extract the properties of the current drawing file. It will open the Inventor Properties dialogue box. (Figure Step 21)

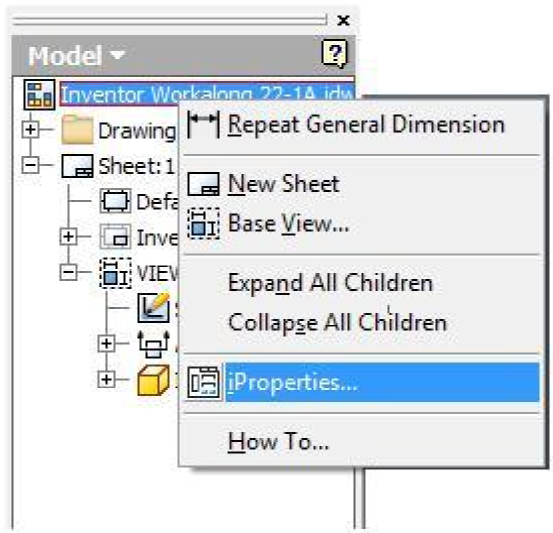


Figure Step 21

Step 22

Enable the Summary tab. Enter TOOL HOLDER – BASE in the Title box and your name in the Author box. (Figure Step 22)

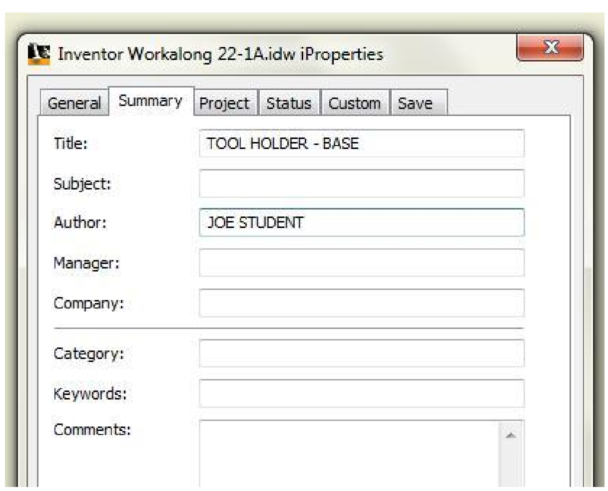


Figure Step 22

Step 23

Enable the Project tab. Enter WORKALONG 25-1A in the Part Number box. Your titleblock should now appear as shown in figure. (Figure Step 23A and 23B)

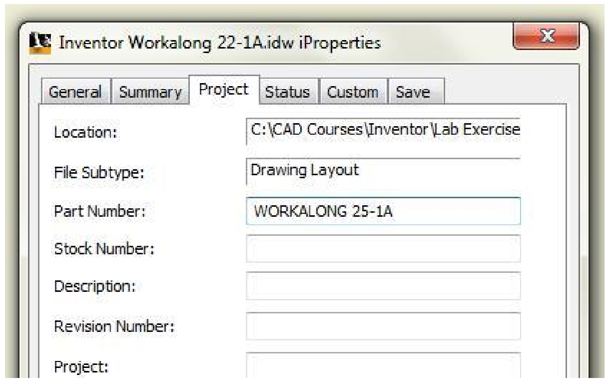


Figure Step 23A

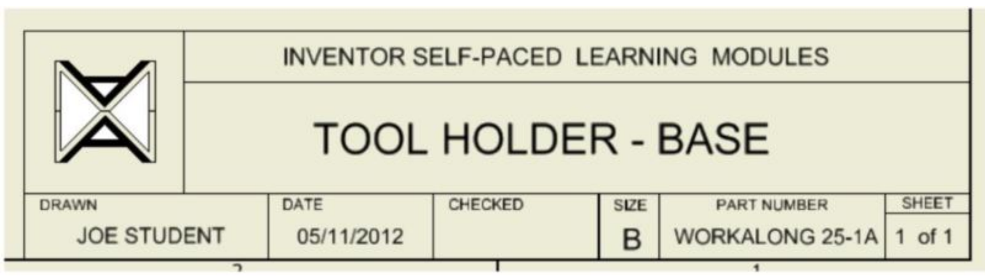


Figure Step 23B

Step 24

Enter the TEXT command and select the location to place the text on the drawing. The cursor will display as a plus sign. Move it to just above the titleblock as shown in the figure. (Figure Step 24)

Figure Step 24

Step 25

When you select the location for the text, the Format Text dialogue box will open. In the text box along the bottom of the dialogue box enter the text as shown in the figure. Ensure that the default text style is set to: Modules Note Text (ANSI) as shown in the dialogue box. (Figure Step 25)

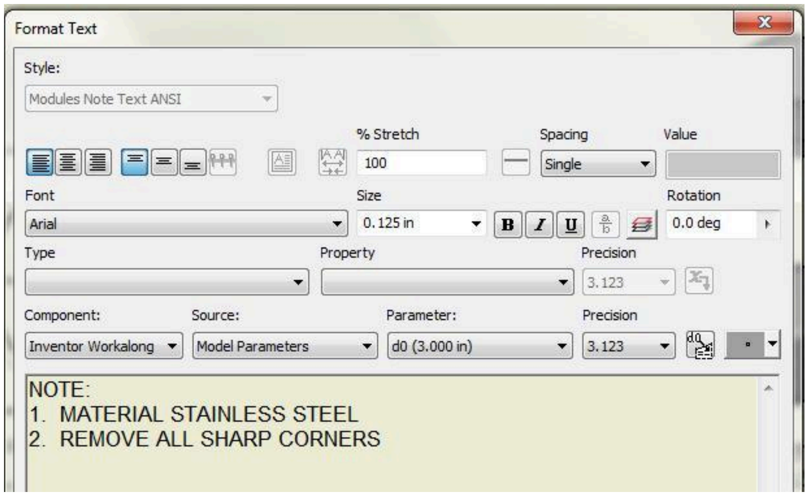


Figure Step 25

Step 26

The completed drawing should appear similar to the figure. (Figure Step 26)

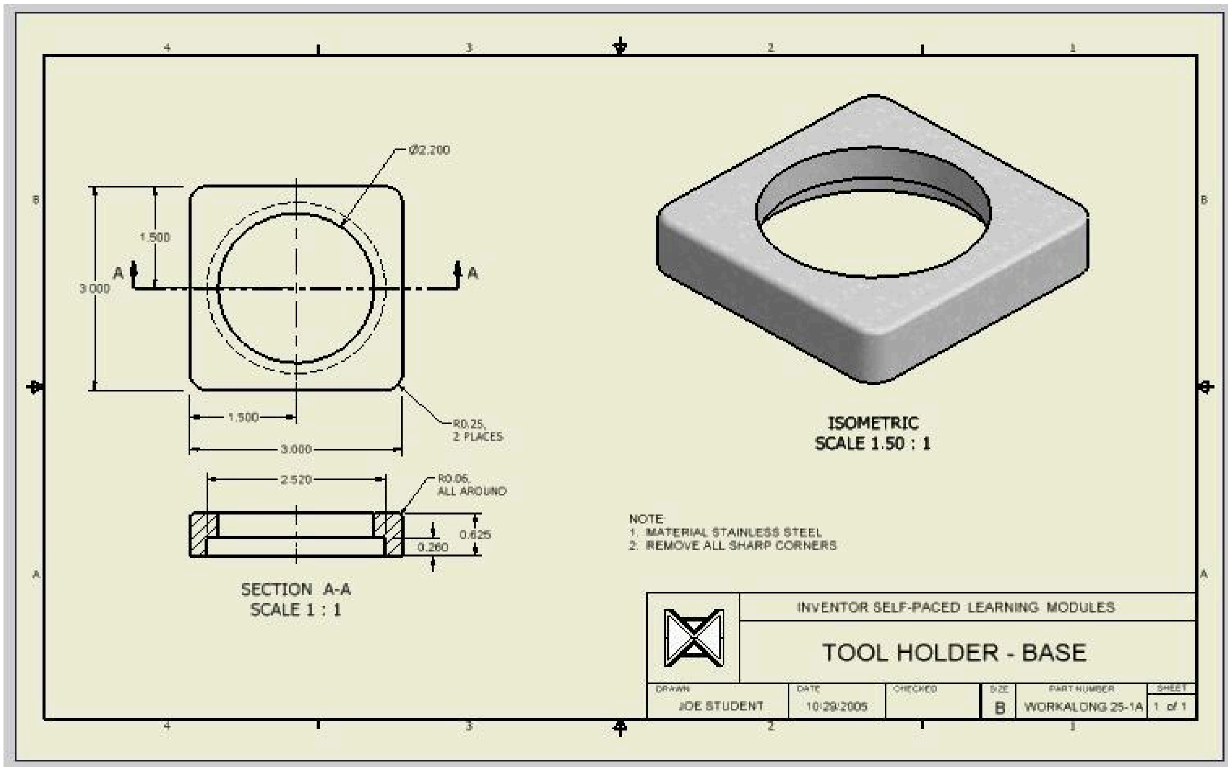


Figure Step 26 [Click to see image full size]

Step 27

Save and close the drawing.

Exporting and Importing Styles

When a style is created in a drawing file it is only useable in that drawing file. To save you re-creating the style in each new drawing file, the style can be saved as a file on the hard drive and then retrieve into another drawing. Saving the style file from a drawing is called exporting and retrieving it into a drawing is called importing.

WORK ALONG: Exporting and Importing Styles

Step 1

Using Windows Explorer, create the folder: Style Library in the existing folder: C:\CAD Courses\ Inventor Course. (Figure Step 1)

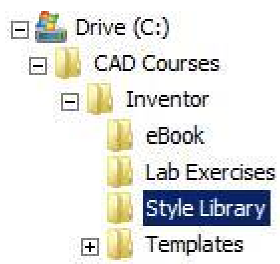


Figure Step 1

Step 2

Start Inventor and check the default project. If necessary, set it to: Inventor Course.

Step 3

Open the drawing: Inventor Lab 24-1A.idw

Step 4

Open the Style and Standard Editor dialogue box. Find and select the style: Modules in (ANSI) under the Dimension heading. It is the dimensioning style that you created earlier in the module. Right-click the name. In the Right-click menu, select Export. (Figure Step 4)

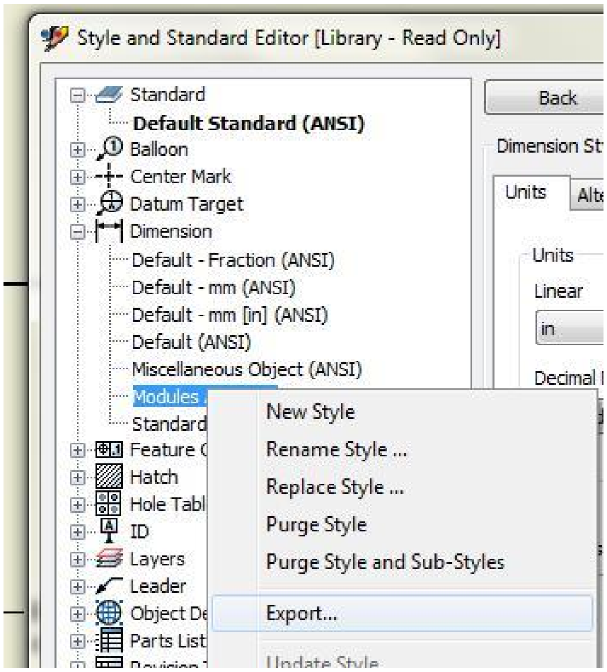


Figure Step 4

Step 5

Select the Style Library folder you created in Step 1. In the File name: box, enter the file name: Modules in (ANSI). (Figure Step 5)

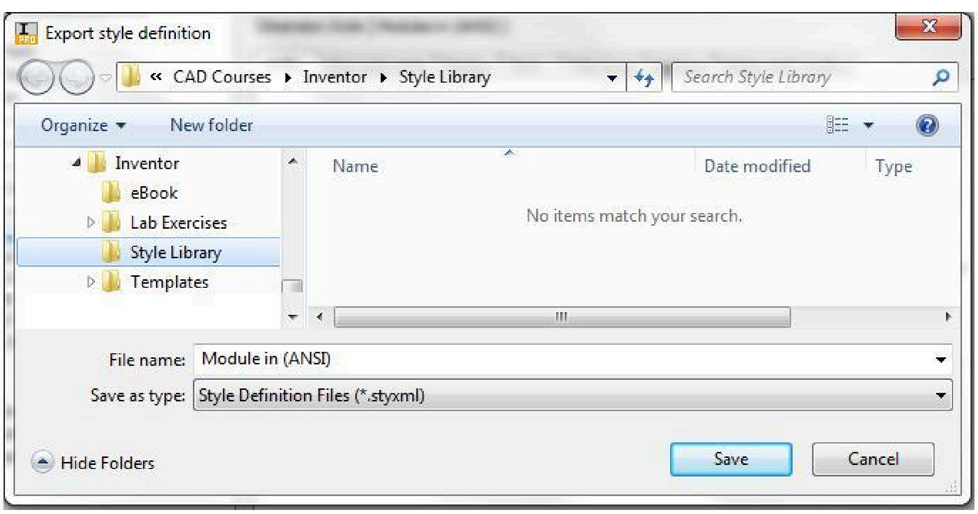


Figure Step 5

Step 6

Open a new drawing file. Enter the STYLE EDITOR command and in the dialogue box, click the Import box located along the bottom of the box. (Figure Step 6)

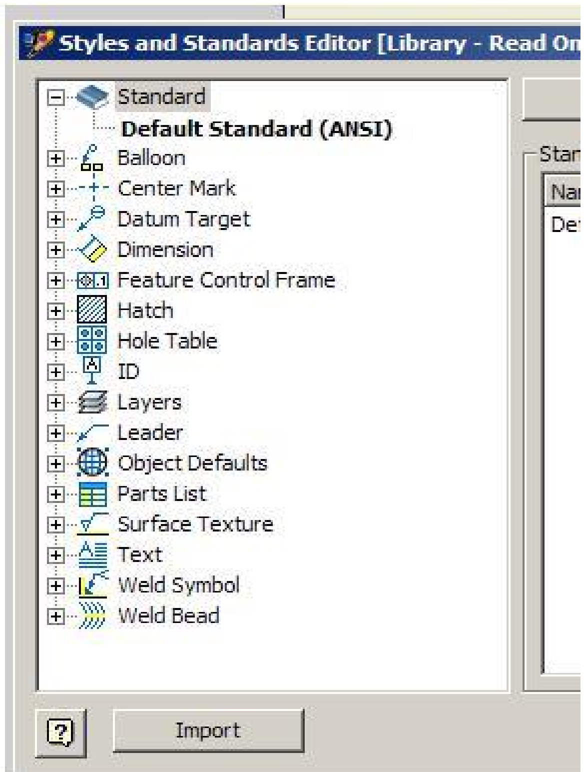


Figure Step 6

Step 7

This will open the Import style definition dialogue box. Click the folder Style Library and select the file: Modules in (ANSI) .styxml that you exported in Step 5. (Figure Step 7)

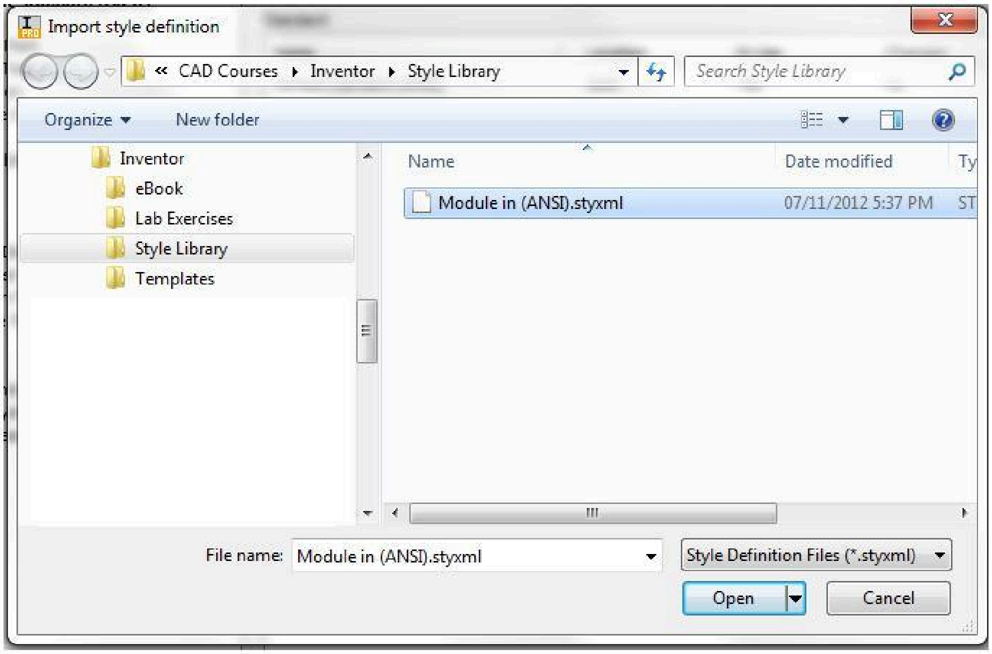


Figure Step 7

Step 8

Activate the file: Inventor Lab 24-1A.idw and export the other two styles that you created in this drawing. Figure Step 8 shows the three styles that should now be in the Style Library folder. You can check it using Windows Explorer. (Figure Step 8)

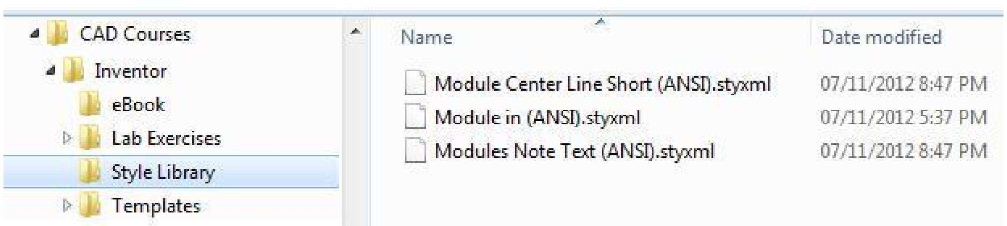


Figure Step 8

Step 9

Open a new drawing file. Enter the STYLE EDITOR command and in the dialogue box click the Import box located along the bottom of the box. This will open the Import style definition dialogue box. This will open the Import style definition dialogue box. With the Look in: box displaying the folder Style Library, select the file: Modules in (ANSI) .styxml that you exported in Step 5. (Figure Step 9)

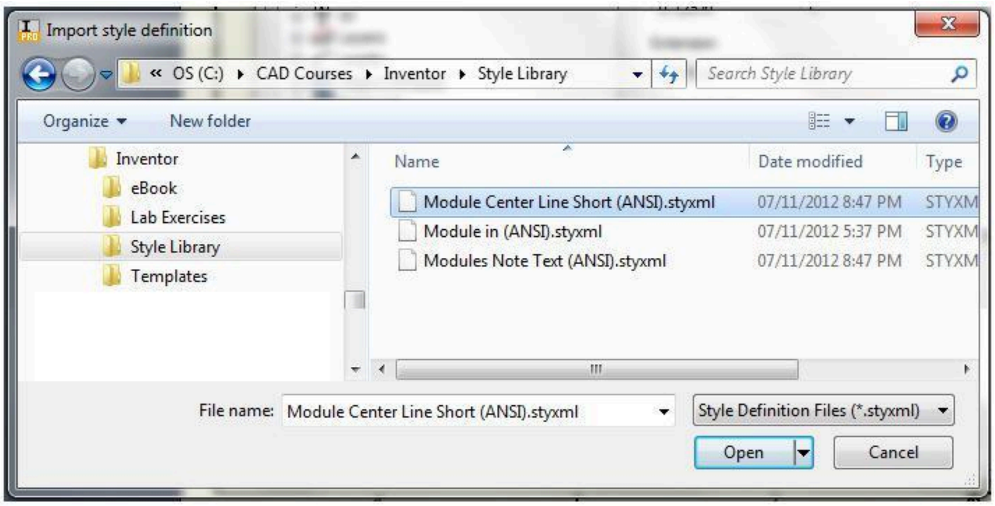


Figure Step 9

Key Principles

Key Principles in Module 25

1. The default styles cannot be renamed nor should they be altered. It is always best to create your own named style using a default style to copy from.
2. A style is a named set of variables or settings that controls the way the annotation will appear on the drawing.
3. The RETRIEVE DIMENSION command is used to retrieve the driven and driving dimensions from the model and display them on the drawing. If the driven dimensions in the original sketches are changed, the model dimensions in the drawing will automatically change.

Lab Exercise 25-1

Part Name	Project	Units	Template	Color	Material
See Below	Inventor Course	Inches	See Below	N/A	N/A

Step 1

Create the drawing shown below. Ensure that you:
A Retrieve as many model dimensions as you can.
B Add the drawing dimensions to complete the drawing as shown below.
C Import the dimension style: Modules in (ANSI) and use it as the dimensioning style in your drawing. Match the drawing shown below.

D Save the drawing files with the drawing name shown below.

Part: Post
Drawing Size: D
Drawing Name: Inventor Lab 25-1.idw
Part Name: Inventor Workalong 22-1B.ipt
Template: English-Modules Drawing ANSI (in).idw (Figure Step 1A, 1B, and 1C)

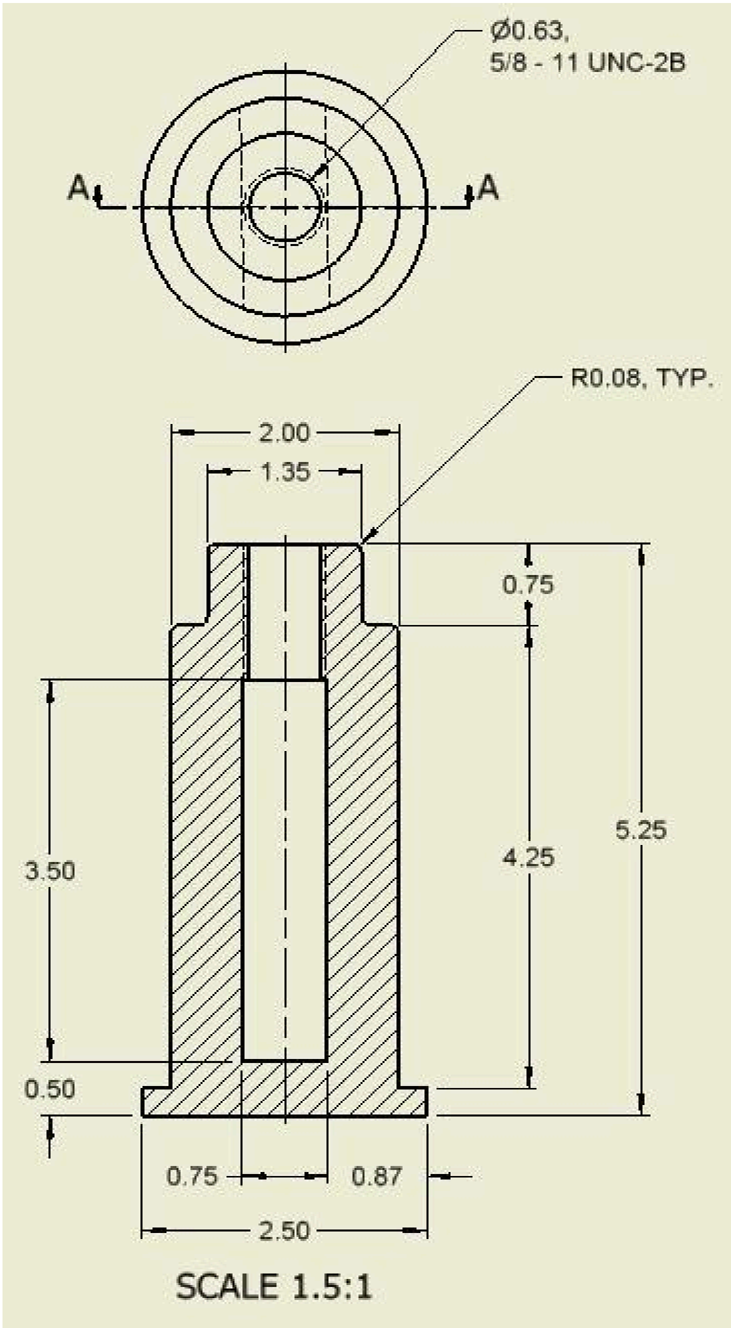


Figure Step 1A

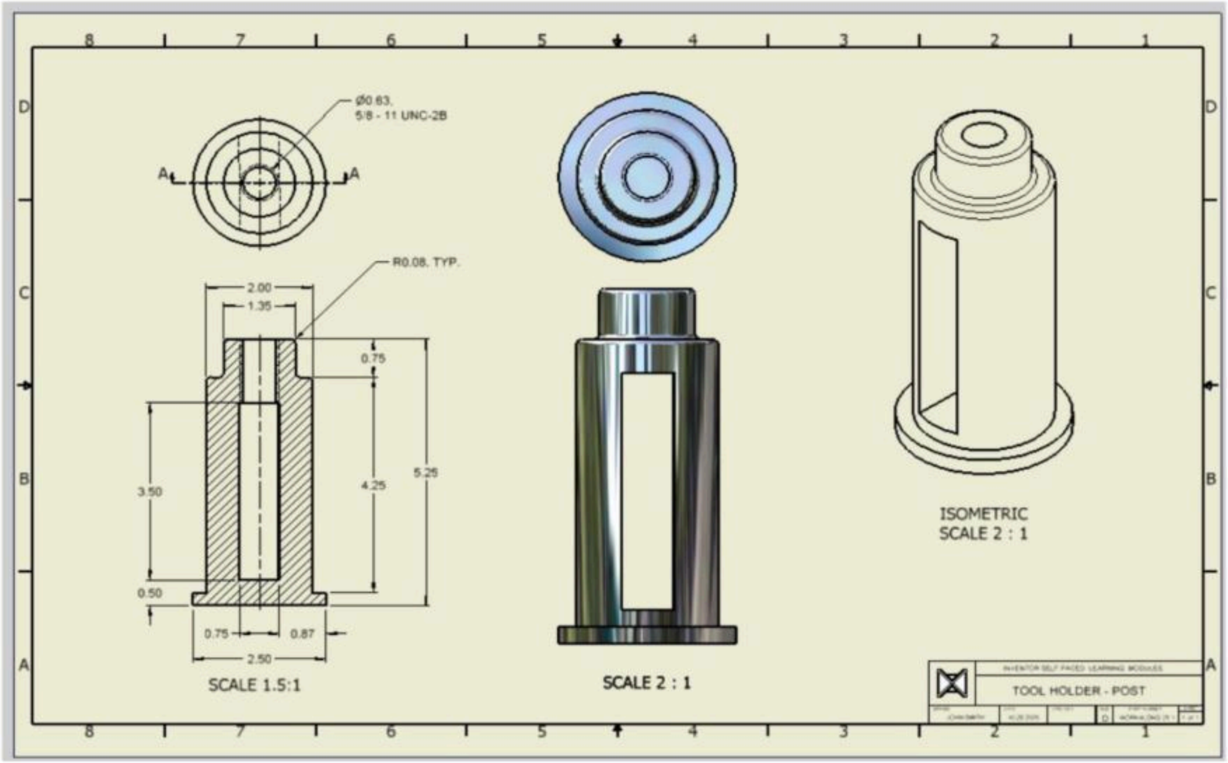


Figure Step 1B [Click to see image full size]

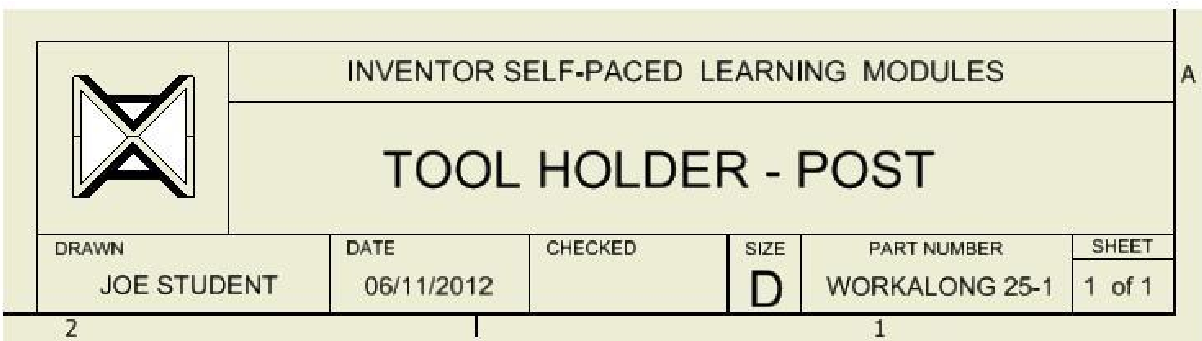


Figure Step 1C

Module 26 Competency Test No. 5 Open Book

Learning Outcomes

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

1. Within a six hour time limit, complete a written exam and the lab exercise.

The Inventor 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 you have completed this module.

Multiple Choice Questions

Select the BEST answer.

1. What file extension does a presentation file have?
   A. .ipt
   B. .idw
   C. .iam
   D. .iaf
   E. .ipn
2. What is the term used in an assembly file that refers to the part files linked to the assembly?
   A. link
   B. location

C. indicator

D. source

E. reference

3. What does the symbol “<<>>” signify in a dimension text?

A. A reference dimension.

B. The actual dimension number is a model or drawing dimension size.

C. The text size.

D. The dimension units of the model or drawing dimension.

E. That there is no dimension.

4. What is the name of first view you create on a drawing that controls the scale, orientation and location of the orthographical views projected from it?

A. Section View

B. Multiview

C. Projected View

D. Base view

E. Orthographic view

5. What command is used to get the model dimensions for a view?

A. GET DIMENSION

B. GENERAL DIMENSION

C. RETRIEVE DIMENSION

D. RECOVER DIMENSION

E. OBTAIN DIMENSION

6. What is the minimum and maximum number of drawing sheets that can exist in a single drawing file?

A. Minimum – 1 Maximum – none

B. Minimum – 1 Maximum – 10

C. Minimum – 0 Maximum – none

D. Minimum – 1 Maximum – 100

E. Minimum – 0 Maximum – 10

7. What file extension does a drawing file have?

A. .ipt

B. .idw

C. .iam

D. .iaf

E. .ipn

8. What is the method of the construction called when an assembly file is created from a series of part files that were previously created and saved in their own .IPT file?

    A. Part Assembly
    B. Bottom-up Assembly
    C. IPT Assembly
    D. Top-down Assembly
    E. Series Assembly

9. What is the term used to describe the distances that the parts are moved from the grounded part in an explode assembly file?

    A. Distance
    B. Exploded
    C. Trails
    D. Animation
    E. Tweak

10. What file extension does an assembly file have?

    A. .ipt
    B. .idw
    C. .iam
    D. .iaf
    E. .ipn

Lab Exercise 26-1

Time allowed: 6 hours.

Part Name	Project	Units	Template	Color	Material
Inventor Lab 26-1	Inventor Course	Inches	See Below	See Below	See Below

Step 1

Create the following parts. Each part must have its own file and constructed as follows:

A Project the Center Point onto the Base plane.
B Select your own location for X0Y0Z0.
C Draw the necessary sketches and extrude or revolve them to produce the solid models. Apply all of the necessary geometrical and dimensional constraints to maintain the objects shape and size. All sketches must be fully constrained.
D Apply the colour and material shown.
E Ensure that you draw each part in the correct orientation so that they can be easily assembled together. The Home view for each part will help you. (Figure Step 1A, 1B, and 1C)

Figure Step 1A
Solid Model – Home View

Figure Step 1B
Solid Model – Orbited View

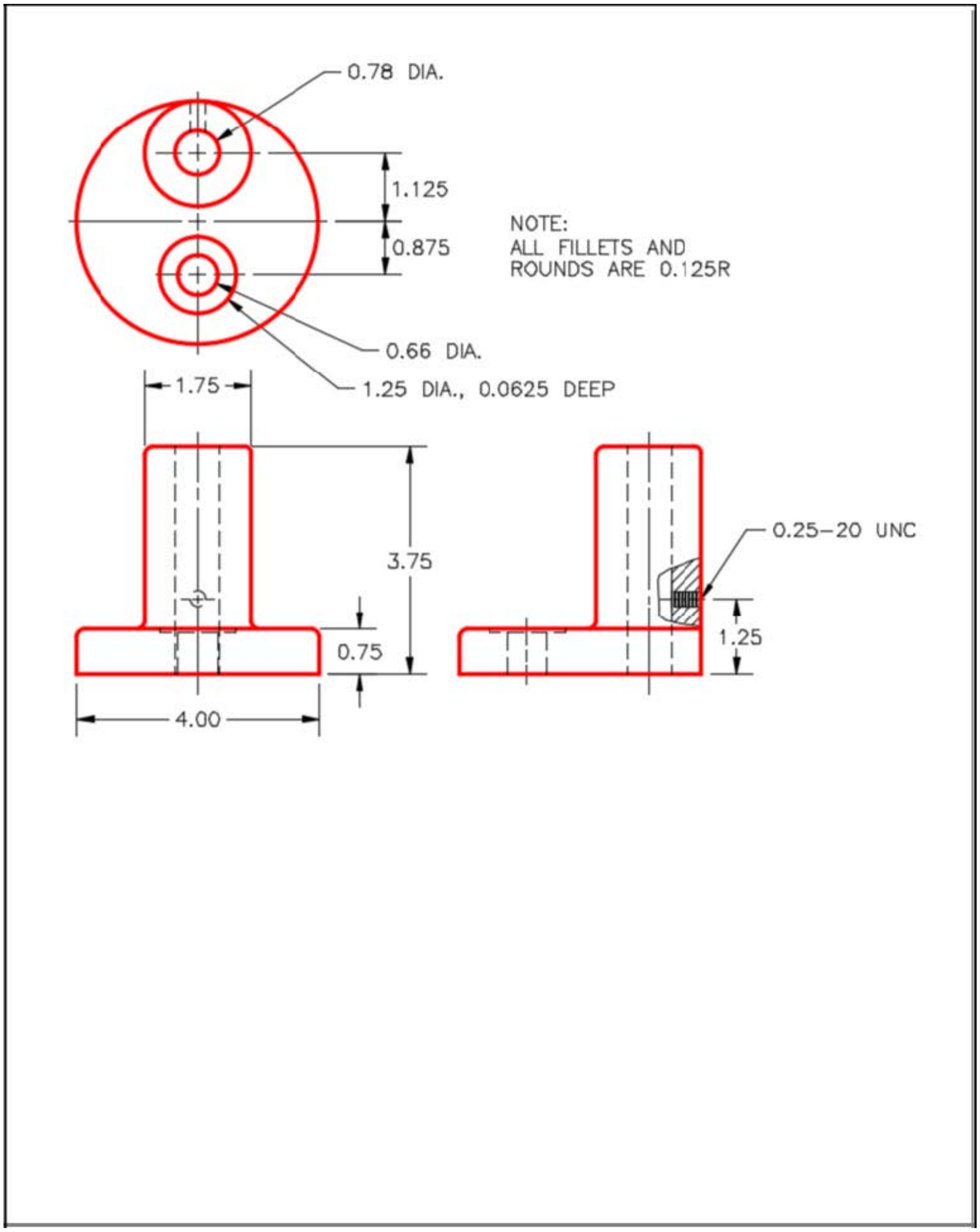


Figure Step 1C
Dimensioned Multiview Drawing [Click to see image full size]

Part: Base
Part Name: Inventor Lab 26-1A
Template: English-Modules Part (in).ipt
Color: Steel – Polished
Material: Stainless Steel

Part: Adjustment Shaft
Part Name: Inventor Lab 26-1B
Template: English-Modules Part (in).ipt
Color: Brass – Satin
Material: Soft Yellow Brass (Figure Step 1D, 1E, and 1F)

Figure Step 1D Solid Model – Home View

Figure Step 1E Solid Model – Orbited View

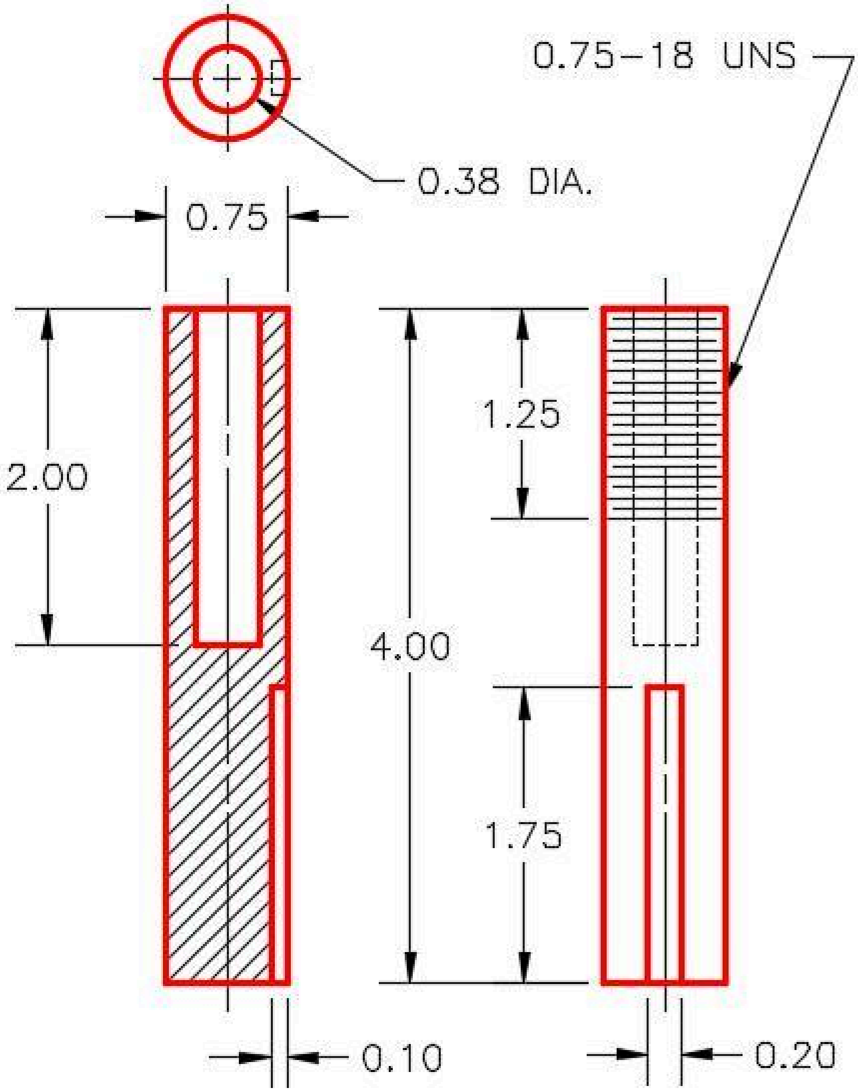


Figure Step 1F
Dimensioned Multiview Drawing

Part: Adjusting Nut
Part Name: Inventor Lab 26-1C
Template: English-Modules Part (in).ipt
Color: Nickel
Material: Non-Alloy Steel (Figure Step 1G and 1H)
Note: Knurl the outside of the nut.

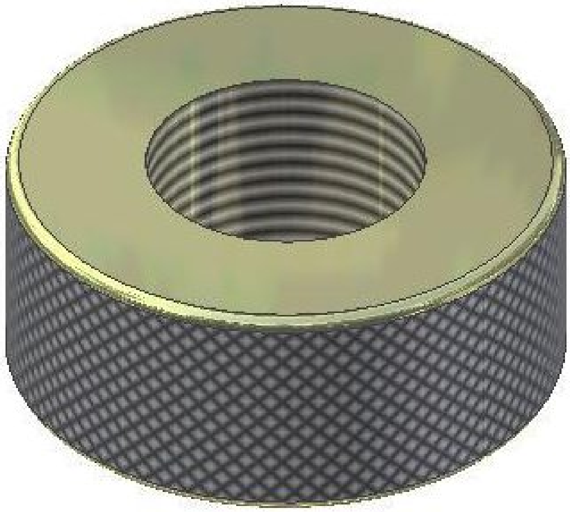


Figure Step 1G
Solid Model – Home View

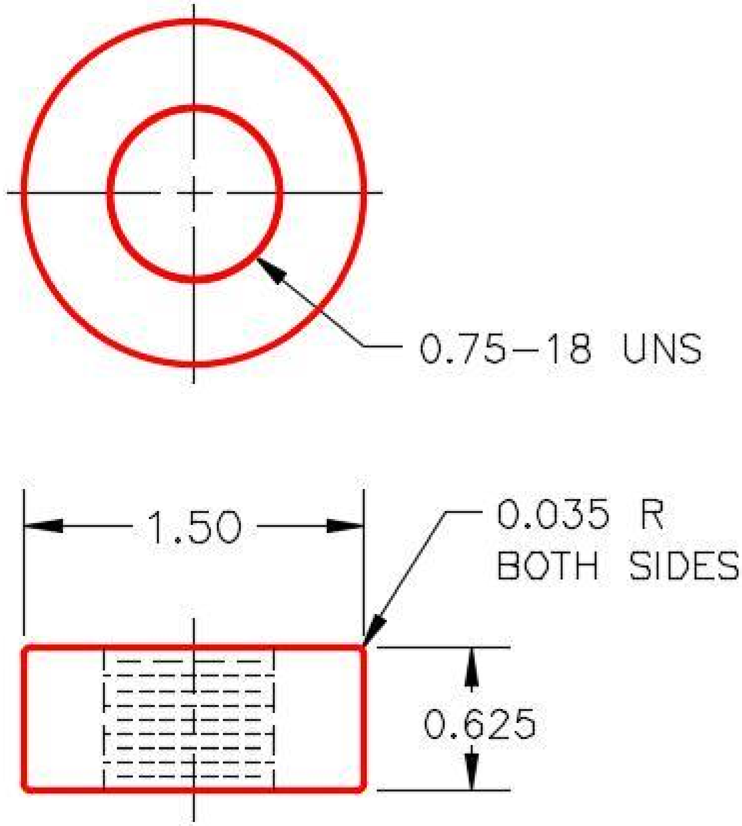


Figure Step 1H
Dimensioned Multiview Drawing

Part: V-Shaft
Part Name: Inventor Lab 26-1D
Template: English-Modules Part (in).ipt
Color: Chrome – Polished Blue
Material: Stainless Steel (Figure Step 1J and 1K)

Figure Step 1J Solid Model – Home View

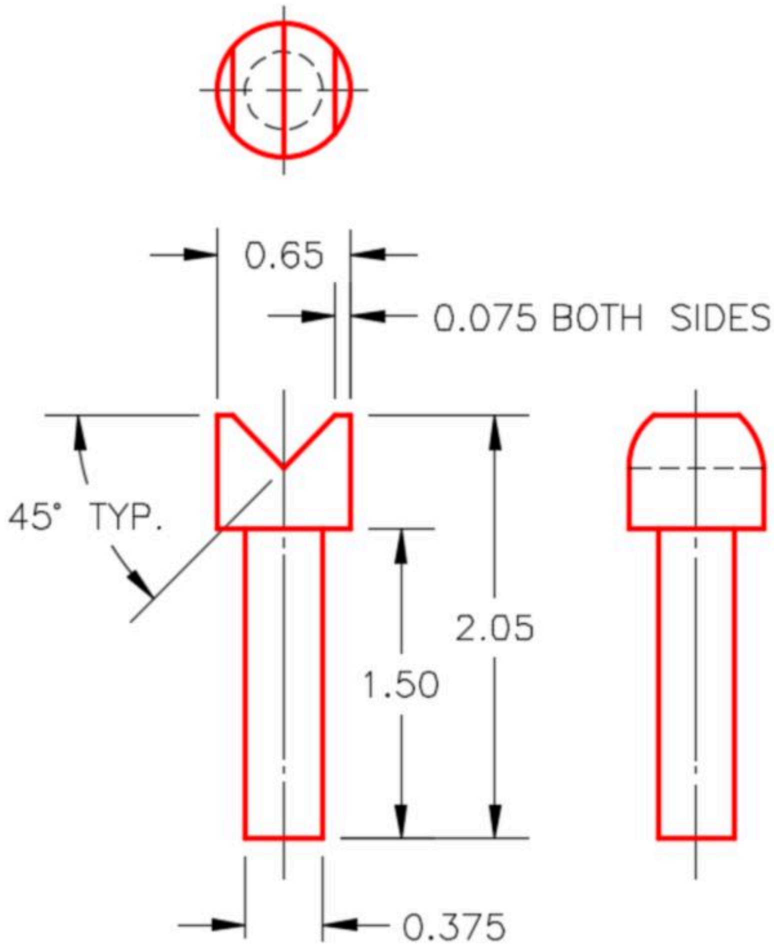


Figure Step 1K
Dimensioned Multiview Drawing

Part: Set Screw
Part Name: Inventor Lab 26-1E
Template: English-Modules Part (in).ipt
Color: Steel – Polished
Material: Steel (Figure Step 1L, 1M, and 1N)

Figure Step 1L
Solid Model –
Home View

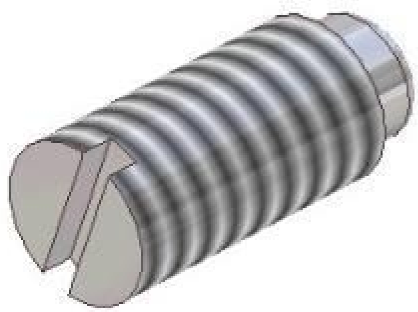


Figure Step 1M
Solid Model –
Orbited View
Figure

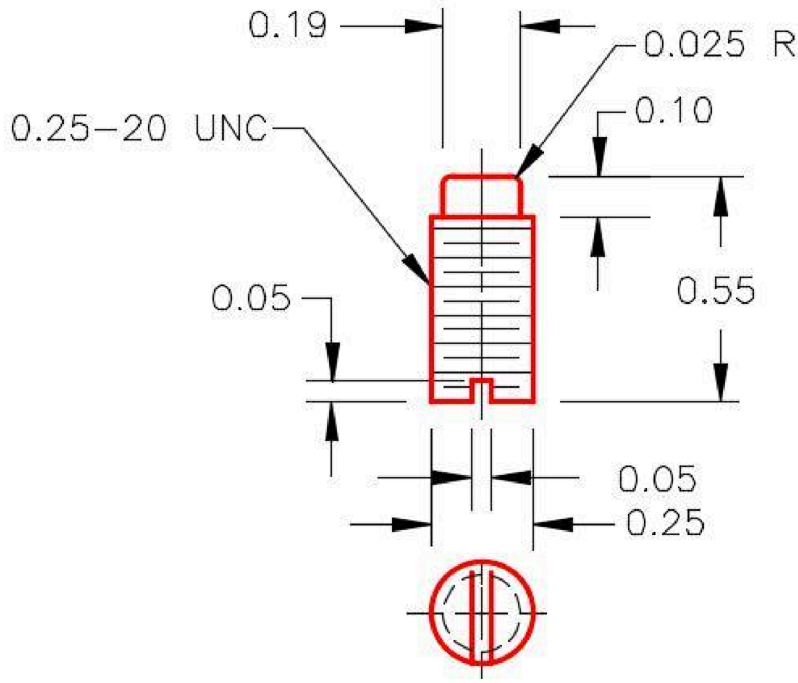


Figure Step 1N
Dimensioned Multiview Drawing

Part: Bolt
Part Name: Inventor Lab 26-1F
Template: English-Modules Part (in).ipt
Color: Metal-AL-6061 – Machined
Material: Steel (Figure Step 1P and 1Q)

Figure Step 1P
Solid Model –
Home View

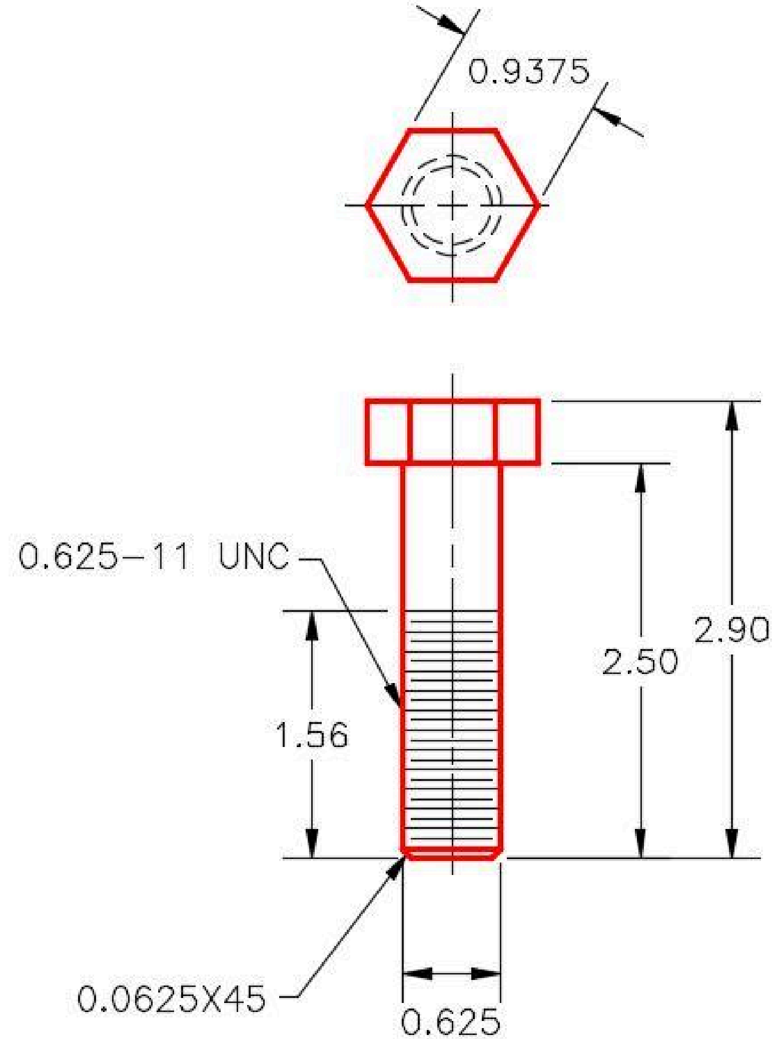


Figure Step 1Q
Dimensioned Multiview Drawing

Part: Washer
Part Name: Inventor Lab 26-1G
Template: English-Modules Part (in).ipt
Color: Metal-AL-6061 – Machined
Material: Steel (Figure Step 1R and 1S)

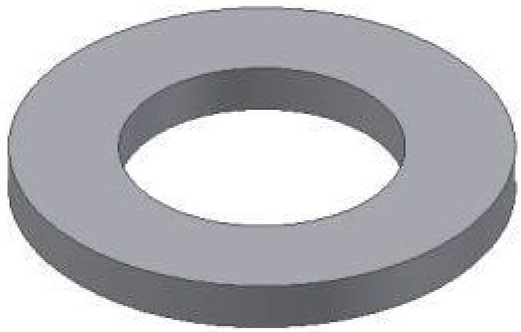


Figure Step 1R
Solid Model –
Home View

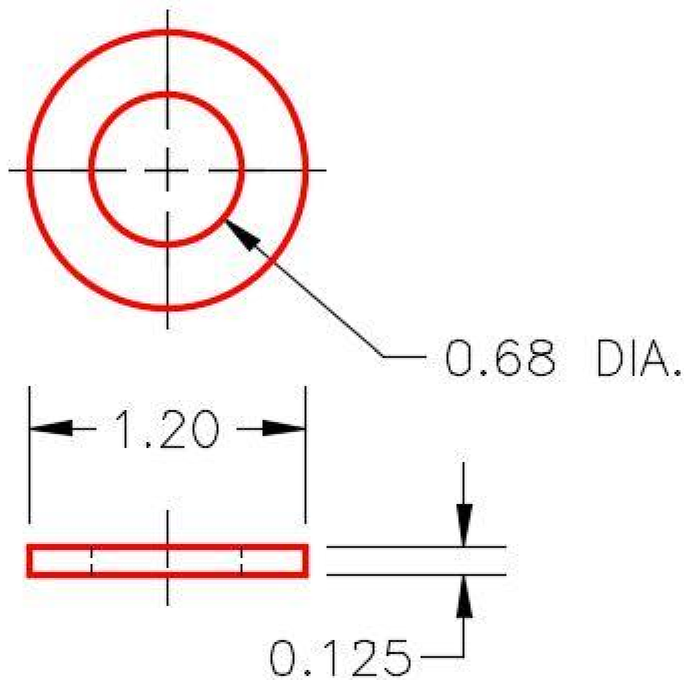


Figure Step 1S
Dimensioned Multiview Drawing

Part: Nut
Part Name: Inventor Lab 26-1H
Template: English-Modules Part (in).ipt
Color: Metal-AL-6061 – Machined
Material: Steel (Figure Step 1T and 1U)

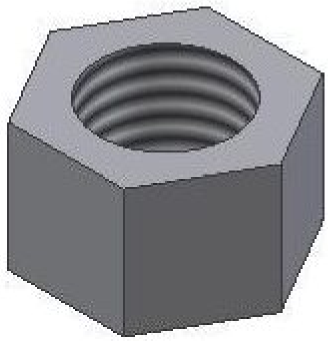


Figure Step 1T
Solid Model –
Home View

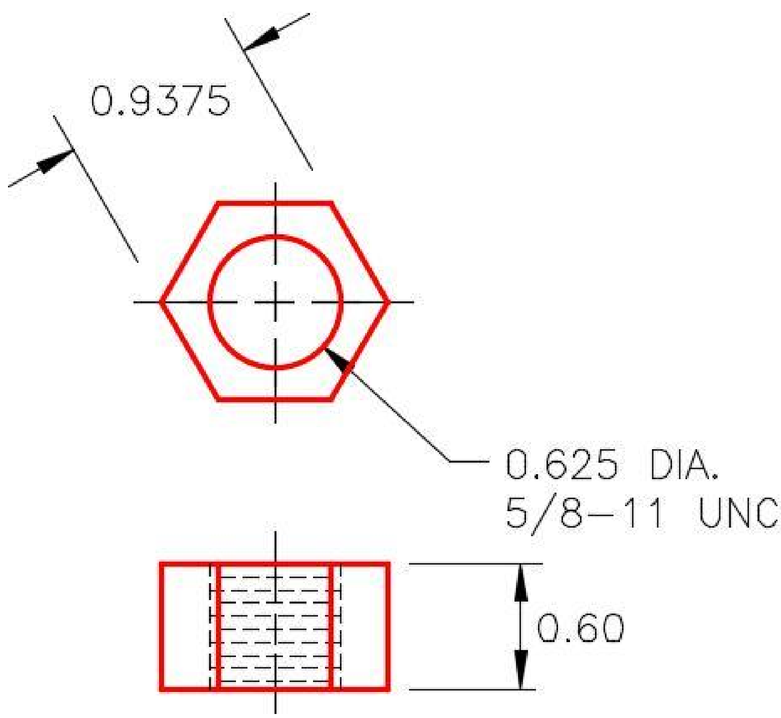


Figure Step 1U
Dimensioned Multiview Drawing
Competency

Step 2

Assemble the parts that you created in Step 1 to create the Machine Jack assembly as shown in figures below. There are two washers, one above and one below the base. The washer under the base should be assembled with 0.5 inches offset from the bottom of the base. Save the assembly using the following name:

Assembly: Machine Jack
Assembly Name: Inventor Lab 26-2A
Template: English-Modules Assembly (in).iam (Figure Step 2A and 2B)

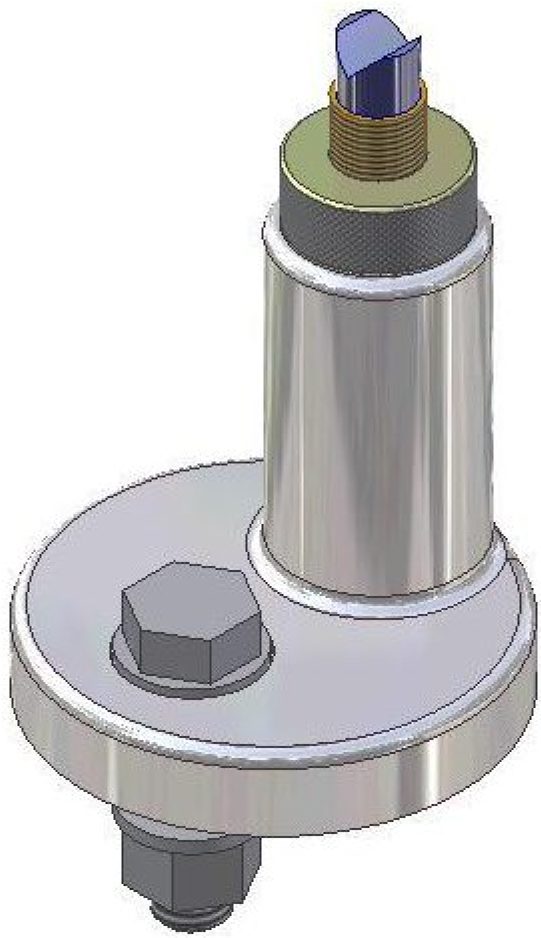


Figure Step 2A
Assembled Solid Model –
Home View

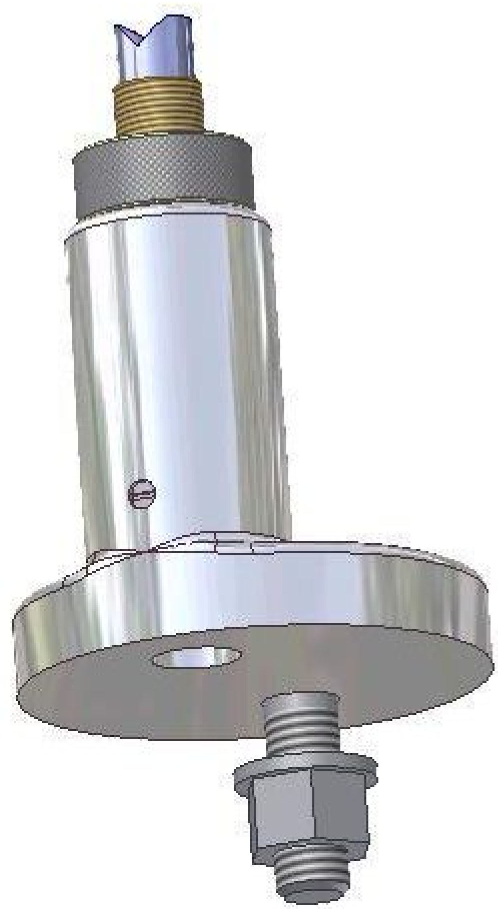


Figure Step 2B
Assembled Solid Model –
Orbited View

Step 3

With the assembly file: Inventor Lab 26-2A.iam as the active file, save a copy of it naming it: Inventor Lab 26-2B.iam.

Step 4

Open the assembly file: Inventor Lab 26-2B.iam that you just copied. Change the colour of the Base and the Adjusting Nut in the assembly file to Clear – Blue as shown in the figures below.

Assembly: Machine Jack
Assembly Name: Inventor Lab 26-2B
Template: N/A (Figure Step 4A and 4B)

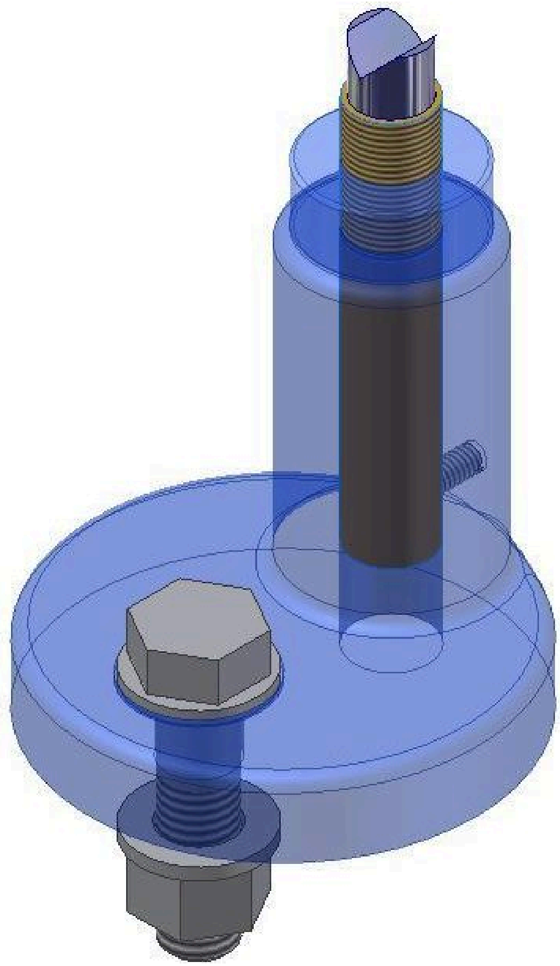


Figure Step 4A
Assembled Solid Model –
Home View

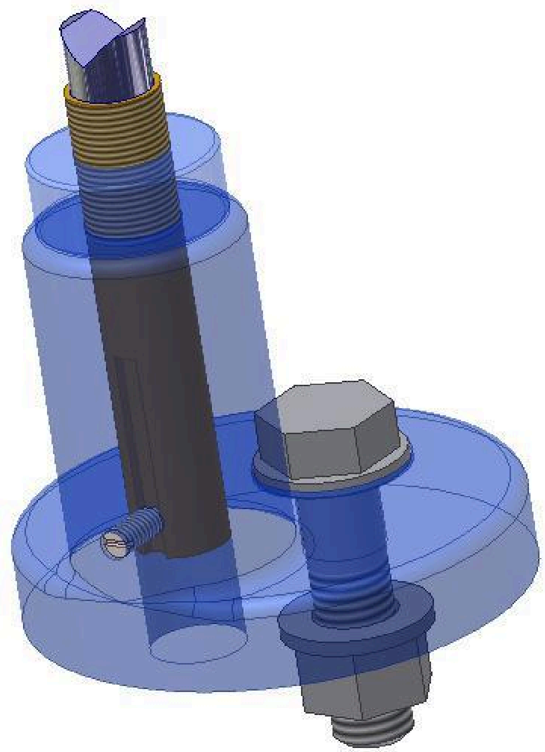


Figure Step 4B
Assembled Solid Model –
Orbited View

Step 5

Create a presentation file using the assembly file Inventor Lab 26-2A. Tweak it to match as close as you can to Figure Step 5.

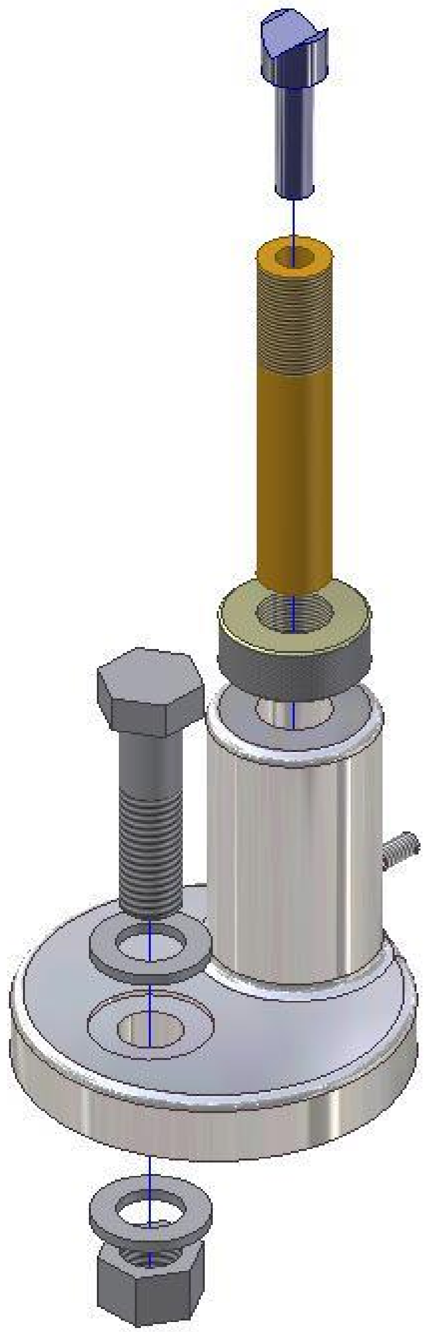


Figure Step 5

Step 6

Set the animation Interval to 20 and the Repetitions to 2. Test the animation.

Presentation: Machine Jack
Presentation Name: Inventor Lab 26-2A
Template: N/A

Step 7

Create the following drawing as follows:

A Create one drawing file complete with two sheets as shown in Figure 4A, 4B and Figure 7A, 7B.
B Create the same views shown in the drawings.
C If the scale is not indicated, set it to full scale or 1:1.
D Save the drawing file with the drawing name: Inventor Lab 26-1.idw Figure Step 5
E Complete the titleblock in both drawings sheets.

Drawing Name: Inventor Lab 26-1.idw

Template: English-Modules Drawing ANSI (in).idw

Sheet: 1
Part: Base
Drawing Size: C
Part Name: Inventor Lab 26-1A.ipt (Figure Step 7A and 7B)

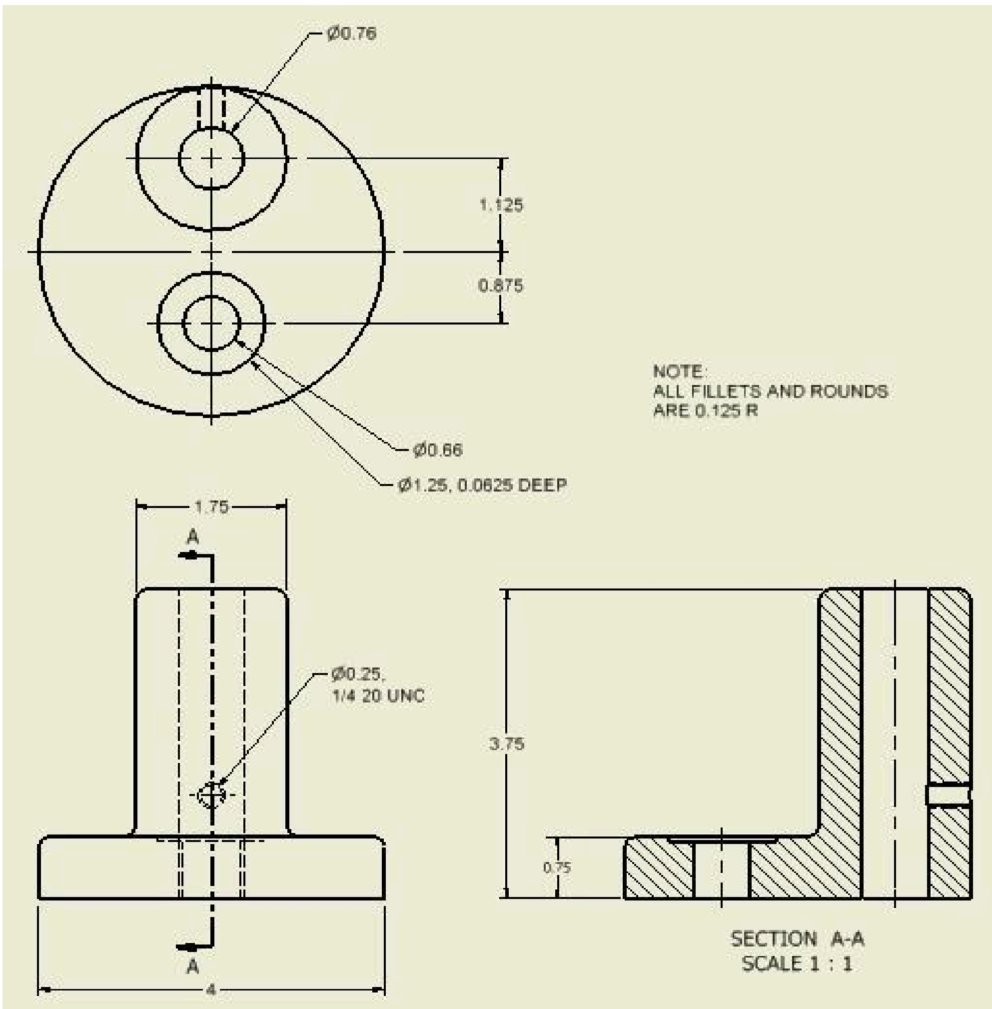


Figure Step 7A [Click to see image full size]

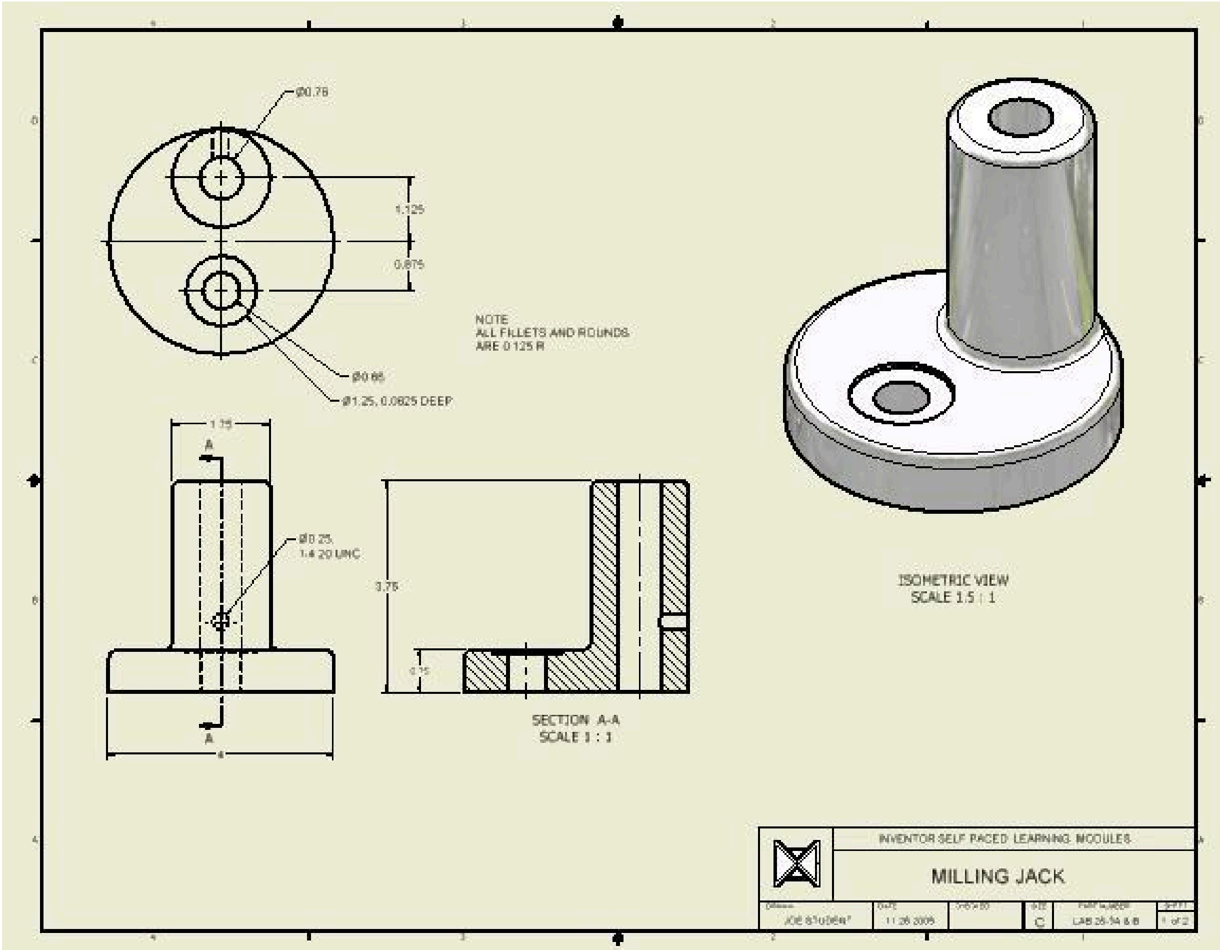


Figure Step 7B [Click to see image full size]

Sheet: 2
Part: Post
Drawing Size: A
Part Name: Inventor Lab 26-1D.ipt (Figure Step 7C and 7D)

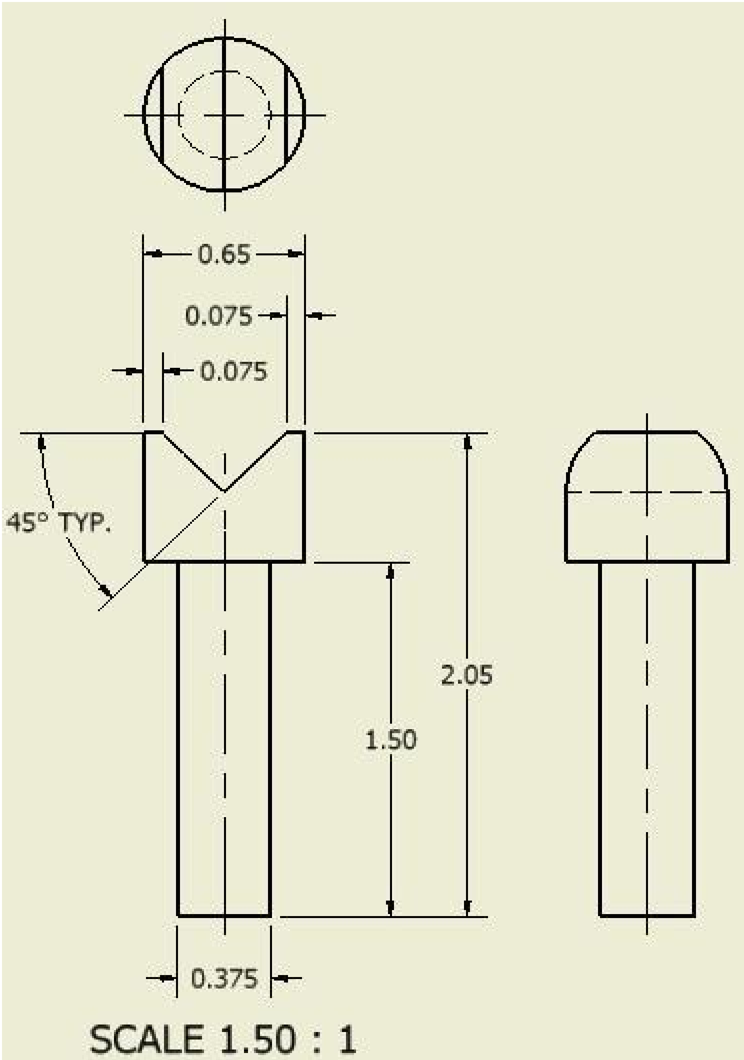


Figure Step 7C

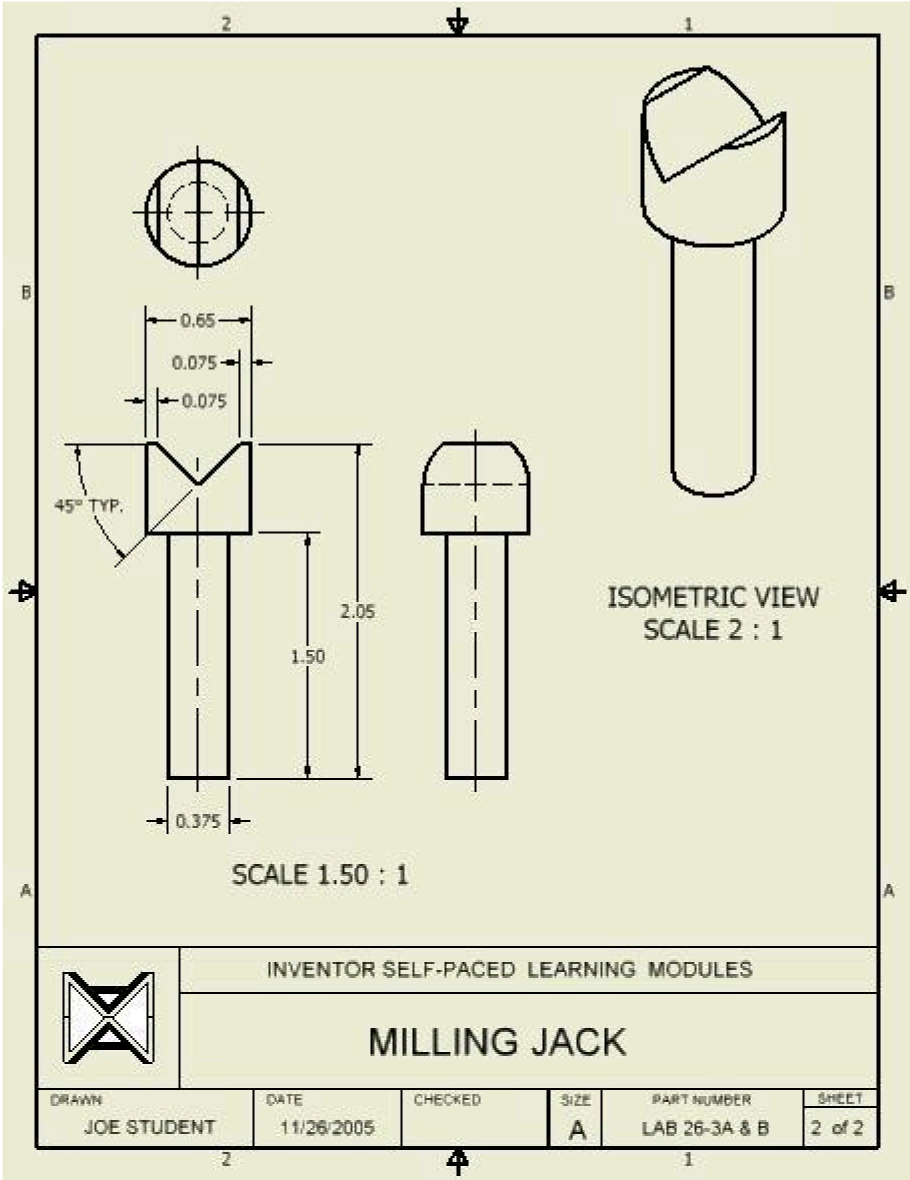


Figure Step 7D

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Autodesk Inventor Module Index

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.	
1.01	August 23, 2023	Fixed typo.	Removed excess letters in "Learning Objectives" text box heading in Module 4 Sketching Lines.