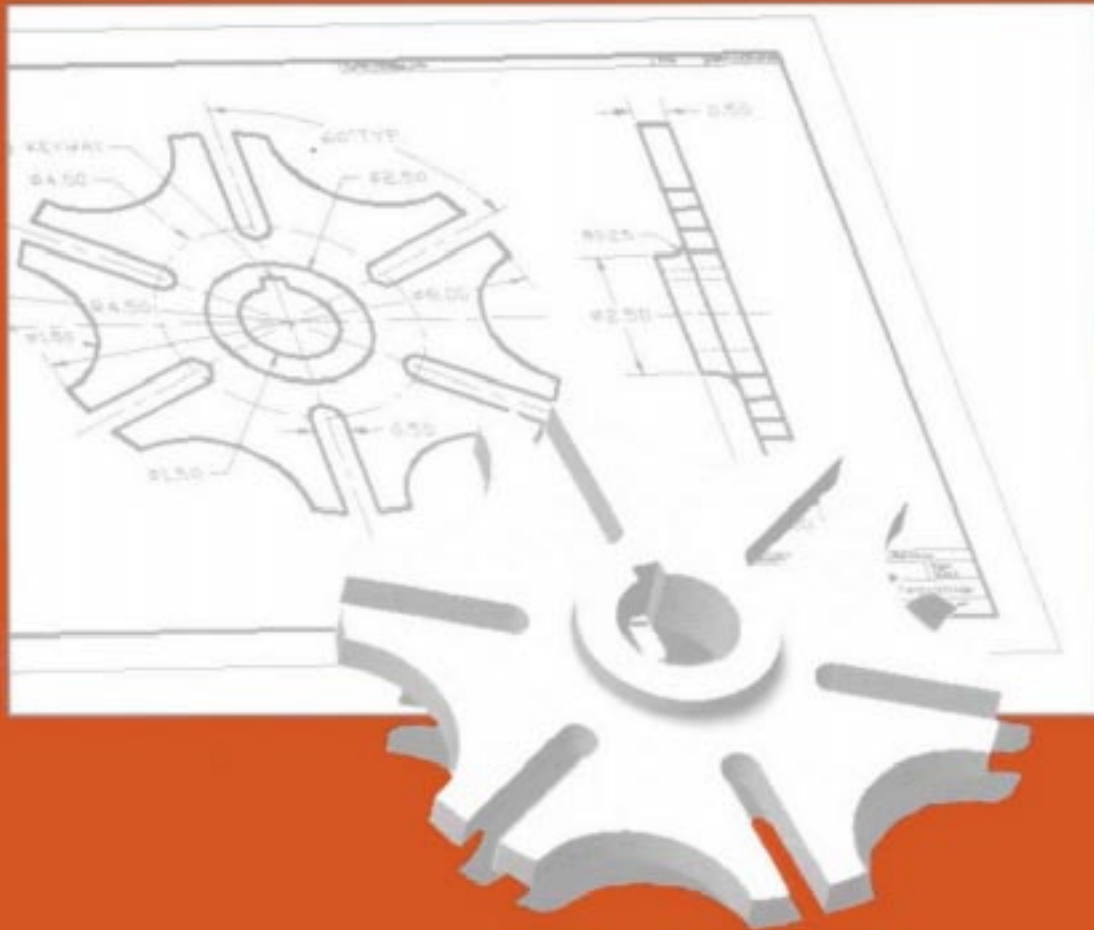


# Parametric Modeling with Mechanical Desktop® 5

A MultiMedia Tutorial



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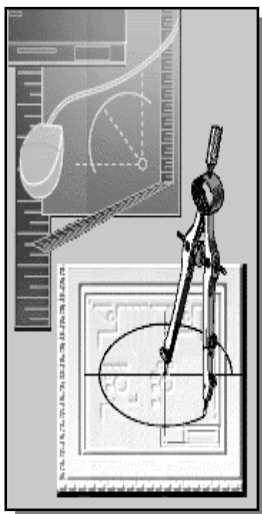
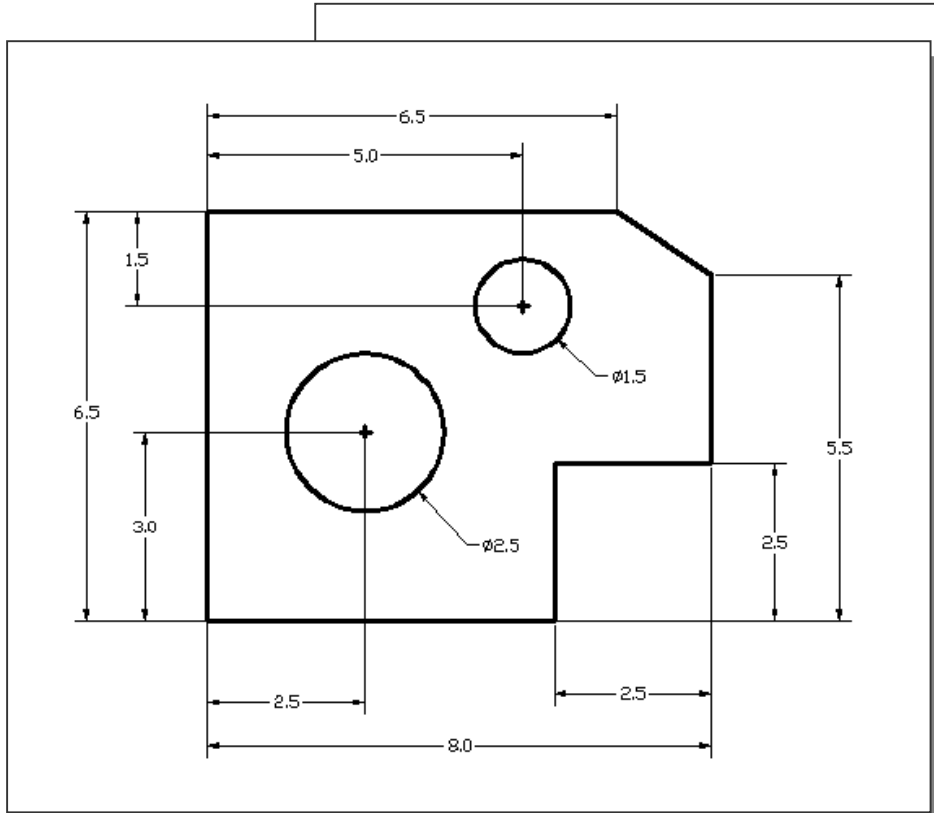
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## Lesson 1

# Parametric Modeling Fundamentals



## Learning Objectives

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

- ◆ Create Simple Parametric Models.
- ◆ Understand the Basic Parametric Modeling Process.
- ◆ Create and Profile Rough Sketches.
- ◆ Understand the "Shape before size" approach.
- ◆ Use the Dynamic Viewing commands.
- ◆ Create and Modify Parametric Dimensions.

## Introduction

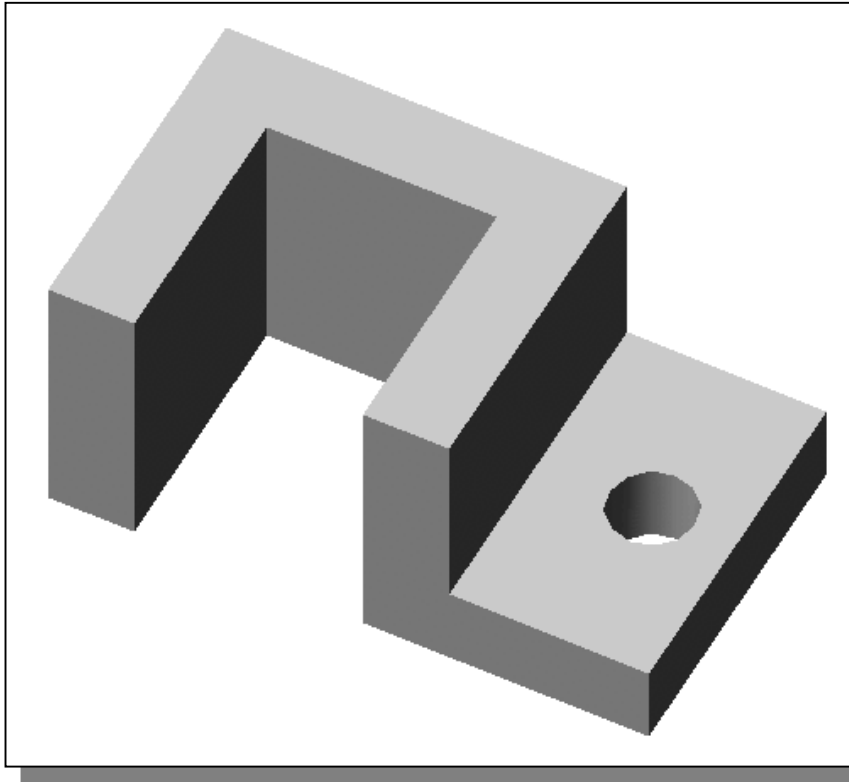
The feature-based parametric modeling technique enables the designer to incorporate the original **design intent** into the construction of the model. Parametric modeling is accomplished by first identifying the key features of the design and controlling the design variables with parametric sketches.

In *Mechanical Desktop*, the parametric part modeling process involves the following steps:

1. **Create a rough two-dimensional sketch of the basic shape of the base feature of the design.**
2. **Turn the sketch into a parametric profile.**
3. **Apply/modify constraints and dimensions to the profile.**
4. **Extrude, revolve, or sweep the parametric profile to create the base solid feature of the design.**
5. **Add additional parametric features by identifying feature relations and complete the design.**
6. **Perform analyses on the computer model and refine the design as needed.**
7. **Create the desired drawing views to document the design.**

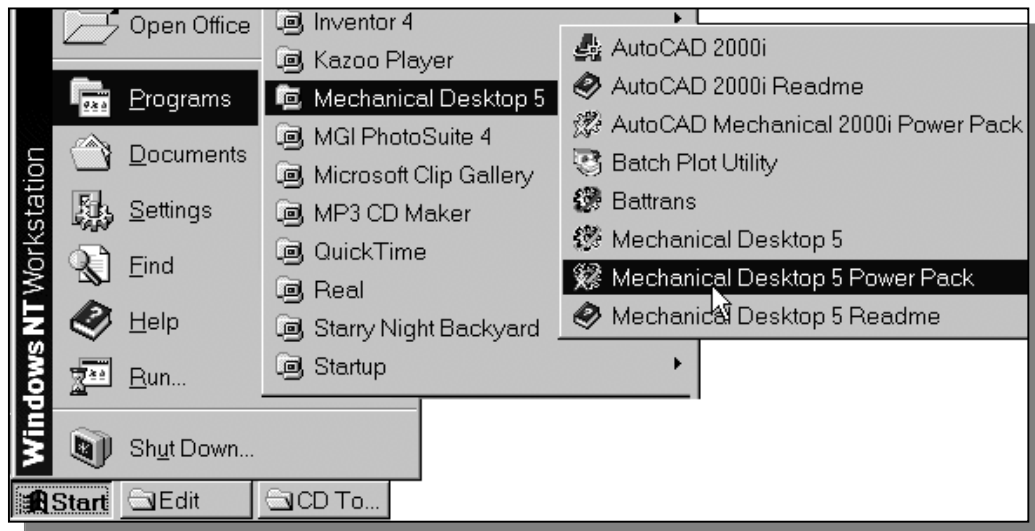
The approach of creating two-dimensional sketches of the three-dimensional features is an effective way to construct solid models. Many designs are in fact the same shape in one direction. Computer input and output devices we use today are largely two-dimensional in nature, which makes this modeling technique quite practical. This method also conforms to the design process that helps the designer with conceptual design along with the capability to capture the design intent. Most engineers and designers can relate to the experience of making rough sketches on restaurant napkins to convey conceptual design ideas. *Mechanical Desktop* provides many powerful modeling and design tools, and there are many different approaches to accomplish modeling tasks. The basic principle of **feature-based modeling** is to build models by adding simple features one at a time. In this chapter, the general parametric part modeling procedure is illustrated; a very simple solid model with extruded features is used to introduce *the Mechanical Desktop* user interface. The use of the *MDT* display viewing functions and the basic two-dimensional sketching tools are also explained and demonstrated.

## The *Adjuster Block* design

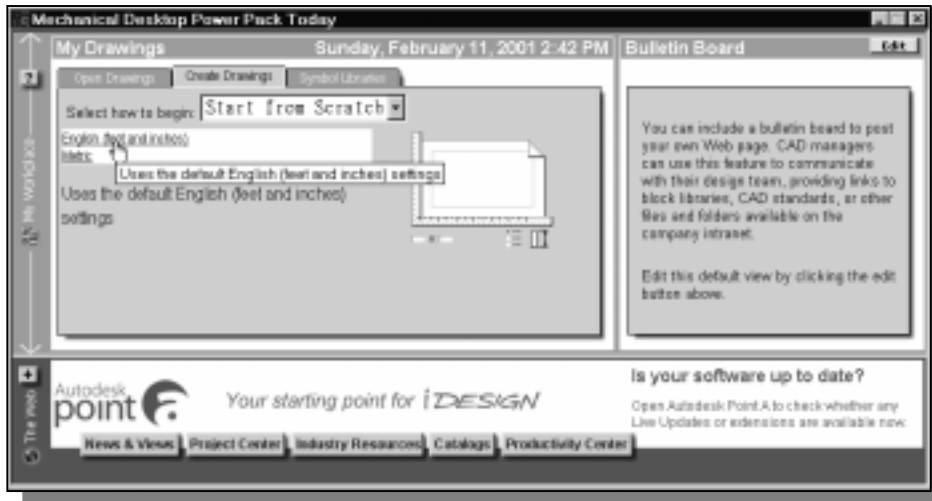


## Starting Mechanical Desktop

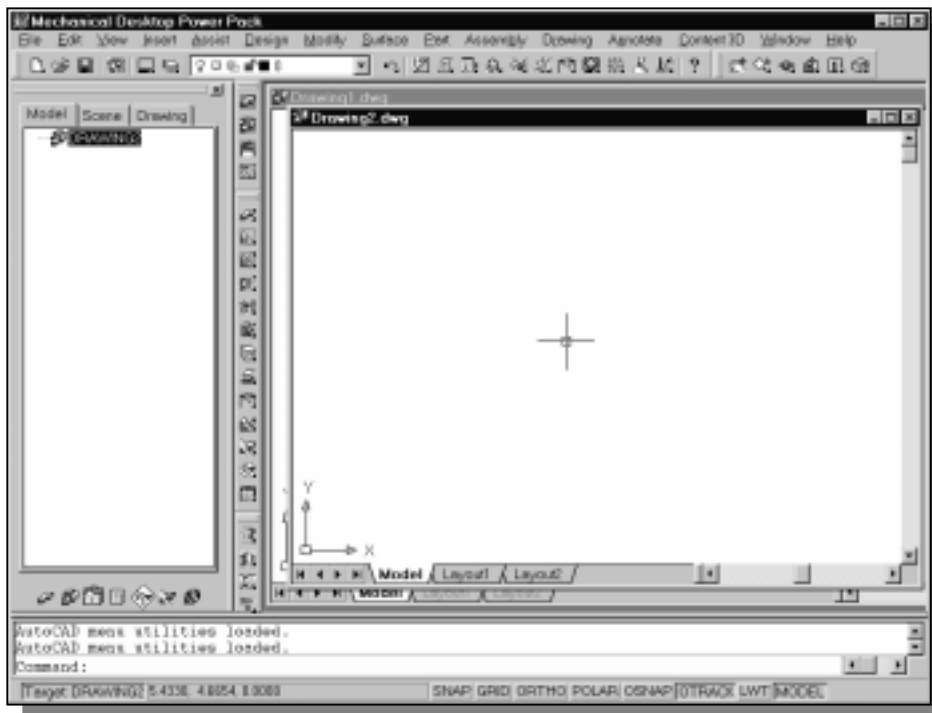
1. Select the **Mechanical Desktop 5 Power Pack** option through the *Start* menu (as shown in the below figure) or select the ***Mechanical Desktop*** icon on the desktop to start *Mechanical Desktop*. The *Mechanical Desktop* main window will appear on the screen.



2. In the *Mechanical Desktop Today* startup dialog box, select the **Create Drawings** tab with a single click of the left-mouse-button.
3. Confirm the startup option is set to *Start from Scratch*, as shown in the figure below.



4. In the *Default Settings* section, pick **English (feet and Inches)** as the drawing units.



- Note that two graphics windows, *Drawing1* and *Drawing2*, are opened. *Mechanical Desktop* automatically assigns generic names, *Drawing X*, as new drawings are created. The first graphics window (*Drawing1*) is created using the default system units. The second graphics window (*Drawing2*) is opened when we chose to create a new drawing from scratch using the default English units.

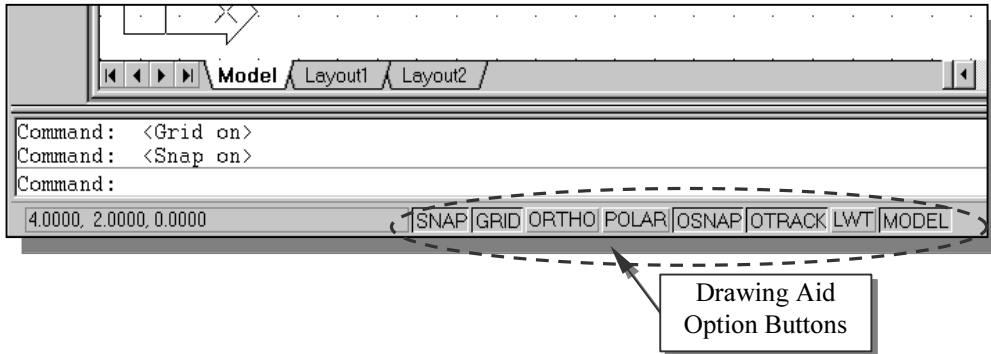
## Creating Rough Sketches

Quite often during the early design stage, the shape of a design may not have any precise dimensions. Most conventional CAD systems require us to input the precise lengths and locations of all geometric entities defining the design, which are not available during the early design stage. With *parametric modeling* systems, we can use the computer to elaborate and formulate the design idea further during the initial design stage. With *Mechanical Desktop*, we can use the computer as an electronic sketchpad to help us concentrate on the formulation of forms and shapes for the design. This approach is the main advantage of *parametric modeling* over conventional solid-modeling techniques.

As the name implies, *rough sketches* are not precise at all. When sketching, we simply sketch the geometry so it closely resembles the desired shape. Precise scale or lengths are not needed. Geometric entities need not be jointed at their ends. Horizontal and vertical lines need not be exactly horizontal or vertical. *Mechanical Desktop* provides us with many tools to assist us in finalizing the sketches. However, if the rough sketches are poor, it will require much more work to generate the desired parametric sketches (*profiles*). Here are some general guidelines for creating sketches in *Mechanical Desktop*:

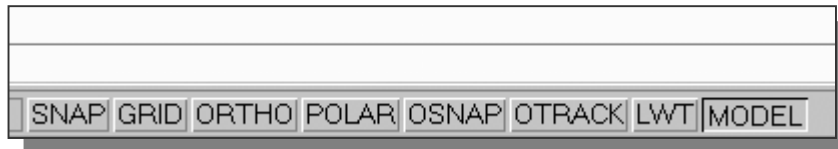
- **Create a sketch that is proportional to the desired shape.** Concentrate on the shapes and forms of the design.
  - **Keep the sketches simple.** Leave out small geometry features such as fillets, rounds and chamfers. They can easily be placed using the fillet and chamfer commands after the parametric sketches have been established.
  - **Exaggerate the geometric features of the desired shape.** For example, if the desired angle is 85 degrees, create an angle that is 50 or 60 degrees. Otherwise, *Mechanical Desktop* might assume the intended angle to be a 90-degree angle.
  - **Draw the geometry so that it does not overlap.** The geometry should eventually form a closed region; small gaps are allowed in the rough sketches.
  - **The gaps in between geometric entities should be smaller than the pickbox size.** *Mechanical Desktop* automatically closes gaps that are smaller than the pickbox size. (The *pickbox* is the small rectangular box that moves with the cursor. The pickbox is used to select geometric entities on the screen.)
- **Note:** The concepts and principles involved in *parametric modeling* are very different, and sometimes they are totally opposite, to those of conventional computer aided drafting. In order to understand and fully utilize *Mechanical Desktop*'s functionality, it will be helpful to take a *Zen* approach in learning the topics presented in this text: **Temporarily forget your knowledge and experiences using conventional Computer Aided Drafting systems.**

## Turning OFF the *Drawing Aid* options

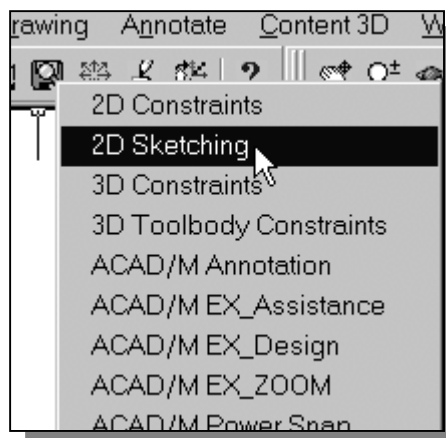


The *Status Bar* area is located at the bottom of the drawing screen. Notice the words SNAP, GRID, ORTHO, POLAR, OSNAP, OTRACK, LWT and MODEL that appear to the right of the *coordinates display*. These are buttons that act as toggle switches; each left-click of the button will toggle the option *ON* and *OFF*. When the corresponding button is *highlighted*, the specific option is turned on. Using the buttons is a quick and easy way to make changes to these *drawing aid* options. We can toggle the options on and off in the middle of another command. The *drawing aid* options are very useful in creating geometry at precise locations and lengths. In *parametric modeling*, the *drawing aid* options can also be used to assure the alignment of parametric features. In this example, to demonstrate the basic principles of *parametric modeling*, we will switch off these options for the first rough sketch.

- In the *Status Bar* area, switch **OFF** all of the *drawing aid* options; leaving only the **MODEL** option switched **ON** as shown below.



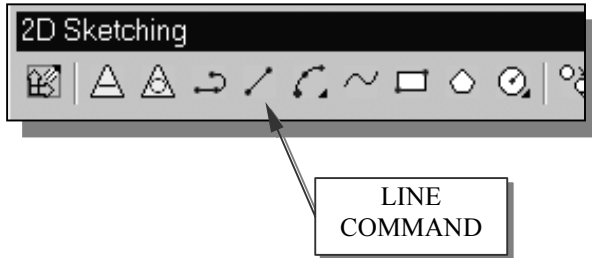
## Displaying the *2D Sketching* Toolbar



1. Move the cursor on any icon in the *Mechanical Main* toolbar (below the pull-down menu) and **right-mouse-click** once to display a list of toolbar menu groups.
2. Select **2D Sketching**, with the left-mouse-button, to display the *2D Sketching* toolbar on the screen.

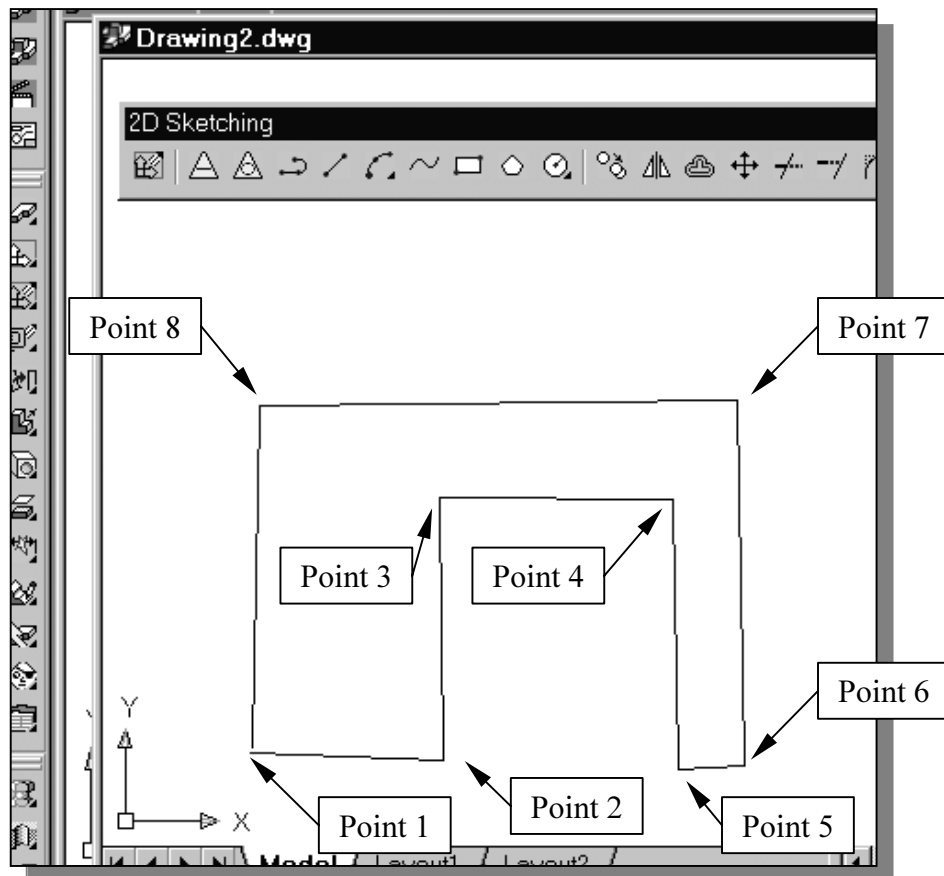


## Step 1: Creating a rough sketch

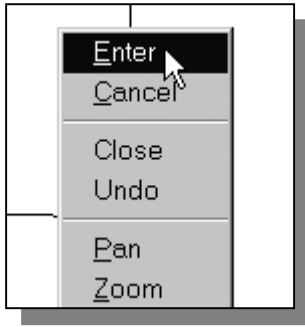


1. Move the graphics cursor to the *Line* icon in the *2D Sketching* toolbar. A *Help-tip* box appears next to the cursor and a brief description of the command is displayed at the bottom of the *Drawing Screen*: “Creates Straight line segments: *LINE*.”

2. Select the icon by clicking once with the **left-mouse-button**; this will activate the *Line* command. In the command prompt area, near the bottom of the *Mechanical Desktop* drawing screen, the message “*\_line Specify first point:*” is displayed. *Mechanical Desktop* expects us to identify the starting location of a straight line.
  3. Move the graphics cursor near the lower-left corner of the drawing screen and create a freehand sketch as shown below. Create the sketch by starting at *Point 1* and ending at *Point 7*, but do not be concerned with the actual size of the sketch. For *Point 7*, select a location that is near *Point 1* and we will intentionally leave a small gap (the gap is smaller than the pickbox size) as shown below.
- ❖ Note that all line segments are sketched nearly horizontally or vertically.



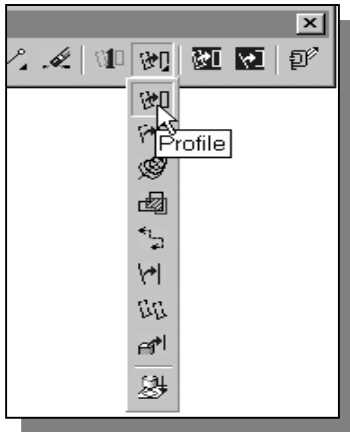




4. Inside the graphics window, click once with the **right-mouse-button** to display the option menu. Select **Enter** in the popup menu to end the *Line* command.

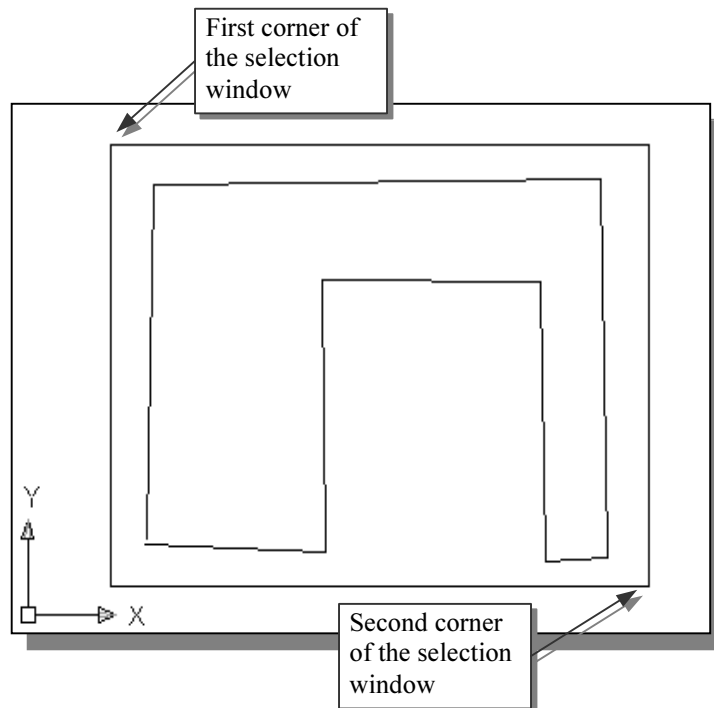
## Step2: Profile the Sketch

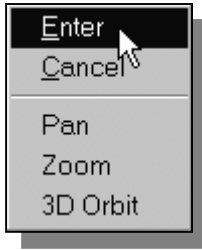
After the geometry is sketched, we will have *Mechanical Desktop* analyze the rough sketch. *Mechanical Desktop* redraws the sketch, with corrections, and specifies how much more information is needed to define the sketch. This operation is used to convert a rough sketch to a parametric profile. This operation is usually referred to as **profiling**.



1. Select **Profile** by left-clicking once on the icon in the *2D Sketching* toolbar. The icon is a picture showing a rough sketch being converted to a *parametric profile*. If a different icon is displayed in the toolbar, press and hold down the left-mouse-button on the icon to display all options and select the **Profile** command from the list.

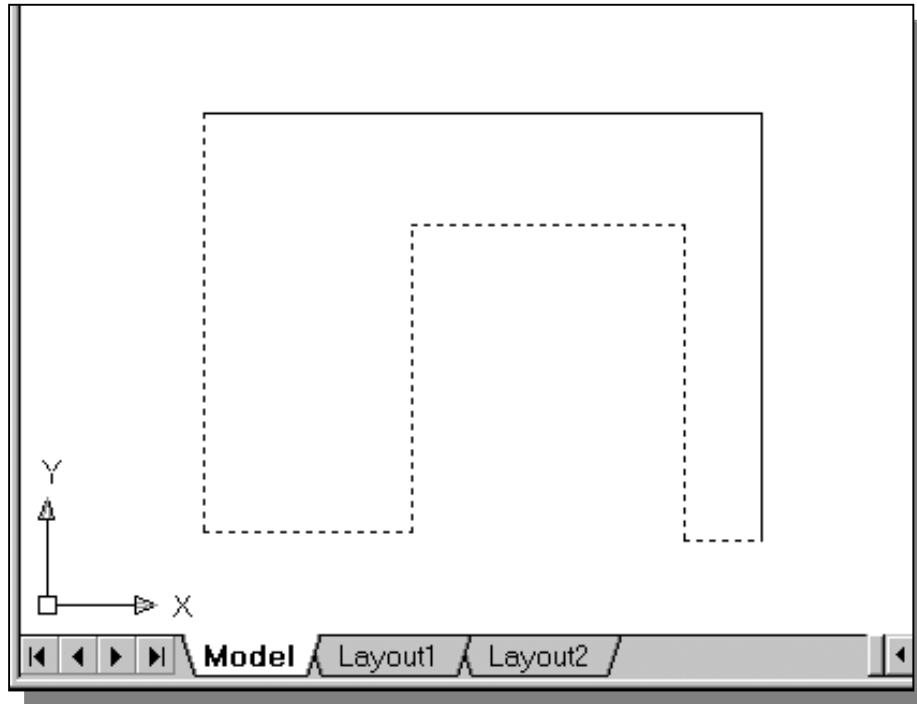
2. The message “*Select objects for sketch:*” is displayed in the command prompt window. Select the rough sketch by enclosing all line segments inside a **selection window**. A *selection window* is a rectangular area that you define in the drawing area by specifying two corner points as shown. (Dragging the window from left to right.)



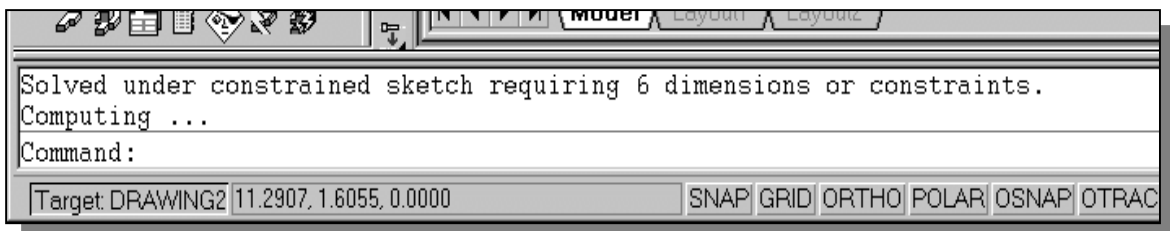


3. Inside the graphics window, **right-mouse-click** once to display the option menu. Select **Enter** in the popup menu to end the selection of entities to be *profiled*.

❖ *Mechanical Desktop* will now analyze and apply the parametric rules to the rough sketch. *Mechanical Desktop* joins the endpoints that are close to each other and adds 2D constraints to straighten lines that are nearly horizontal or vertical. After the geometry is profiled, the number of dimensions or constraints required to fully describe the sketched shape will be displayed in the command prompt area. The nearly vertical or horizontal lines we created are adjusted to vertical and horizontal as shown below.



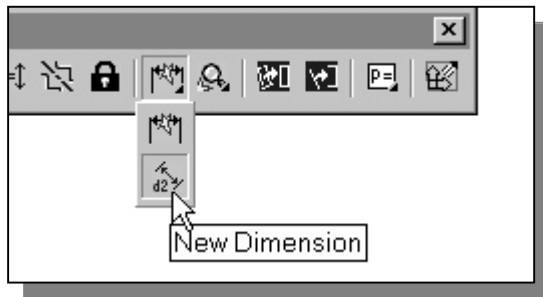
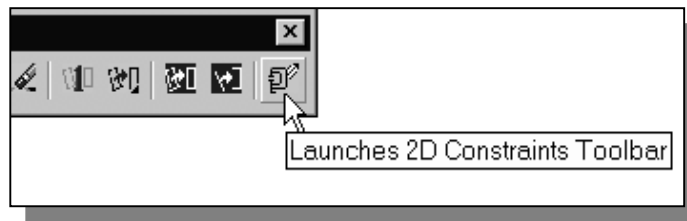
➤ In the command prompt area, *Mechanical Desktop* determined that six additional dimensions or constraints are required to fully describe the sketched geometry. The highlighted entities (the dashed items in the above figure) are used to indicate which geometric items require additional definitions to fully describe the sketch.



### Step 3: Apply/modify constraints and dimensions

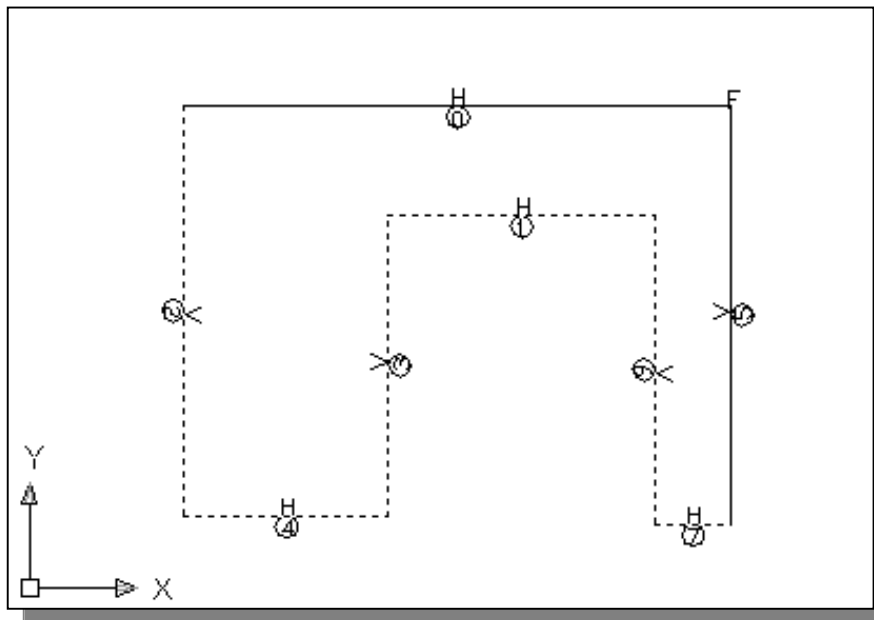
Now that *Mechanical Desktop* has reconstructed and applied some of the geometric constraints (such as horizontal and vertical) to the geometry, we can continue to modify the geometry, apply additional constraints, and define the size of the existing geometry. We will next add dimensions to fully describe the constructed geometry.

1. Move the graphics cursor to the last icon in the *2D Sketching* toolbar. This icon allows us to quickly switch to the *2D Constraints* toolbar. Left-click once on the icon to display the *2D Constraints* toolbar.

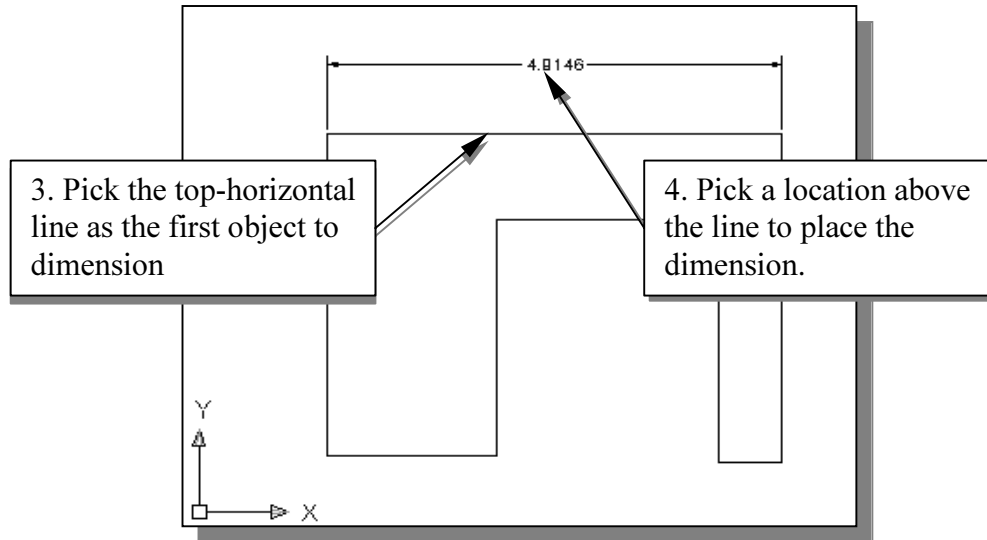


2. In the *2D Constraints* toolbar, select the **New Dimension** command by left-clicking once on the icon.

- Note that the **H** and **V** symbols indicate the *horizontal* and *vertical* constraints applied by the *Mechanical Desktop Profile* command.

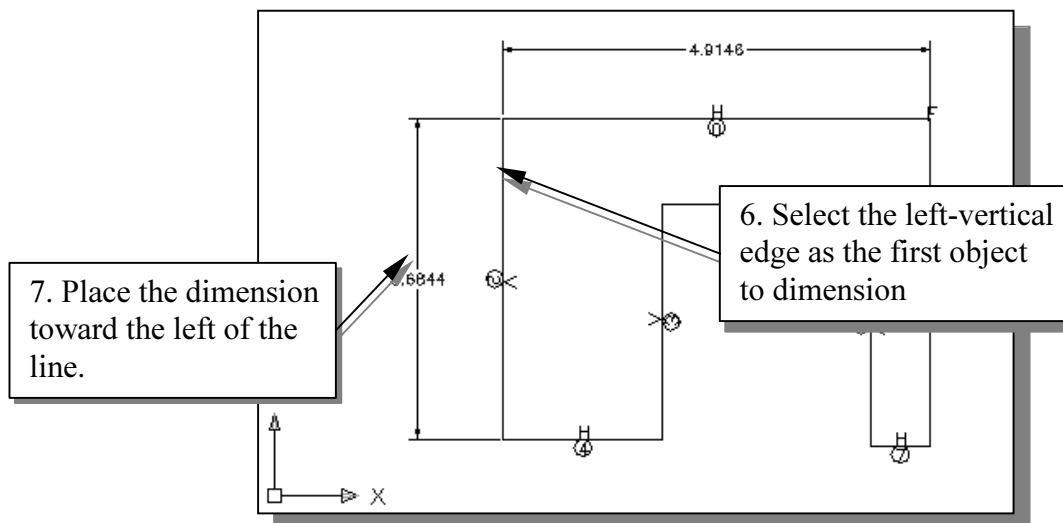


3. The message “*Select first object:*” is displayed in the command prompt window. Select the top-horizontal line by left-clicking once on the line.
4. Move the graphics cursor above the selected line and left-click to place the dimension.

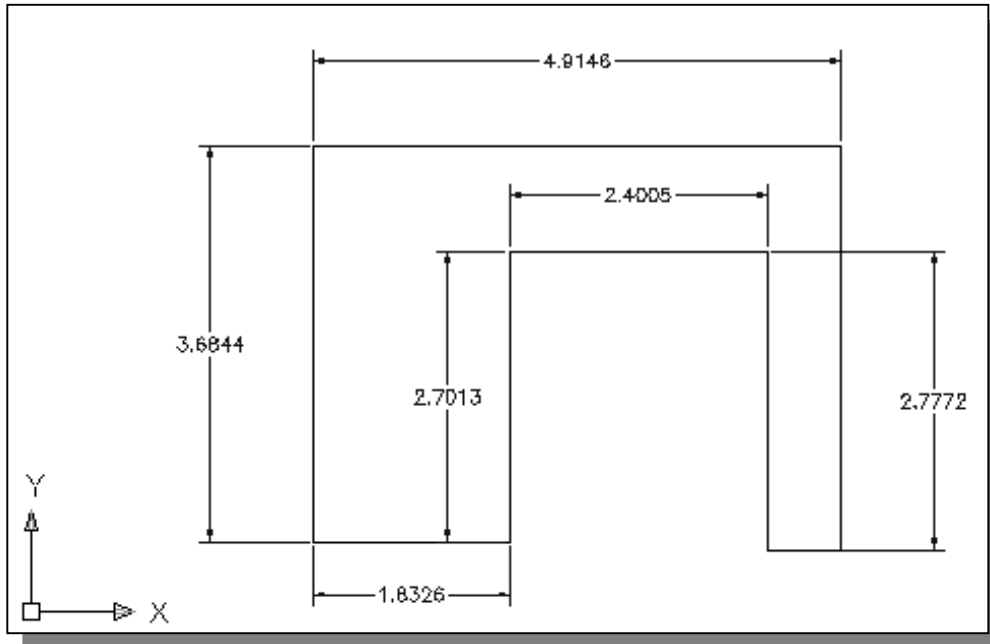


5. In the command prompt area, the message “*Enter Dimension Value or [Undo/Hor..] <4.9146>*” is displayed. Note that the number displayed on your screen may be different. Inside the graphics window, right-mouse-click once and select **Enter** in the popup menu or press the **ENTER** key to accept the displayed value. The dimension now turns green.

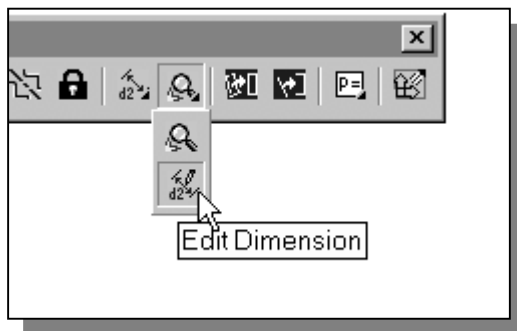
6. The message “*Select first object:*” is displayed in the command prompt window. Select the left-vertical line.
7. Pick a location toward the left of the sketch to place the dimension.



8. In the command prompt area, the message “*Enter Dimension Value or [Undo/Hor..] <3.6844>*” is displayed. Inside the graphics window, right-mouse-click and select **Enter** in the popup menu or press the **ENTER** key to accept the displayed value.
- On you own, repeat the above steps and create additional dimensions so that the sketch appears as shown below. Note that in the command prompt area, the message “*Solved fully constrained sketch*” is displayed after all six dimensions are added.

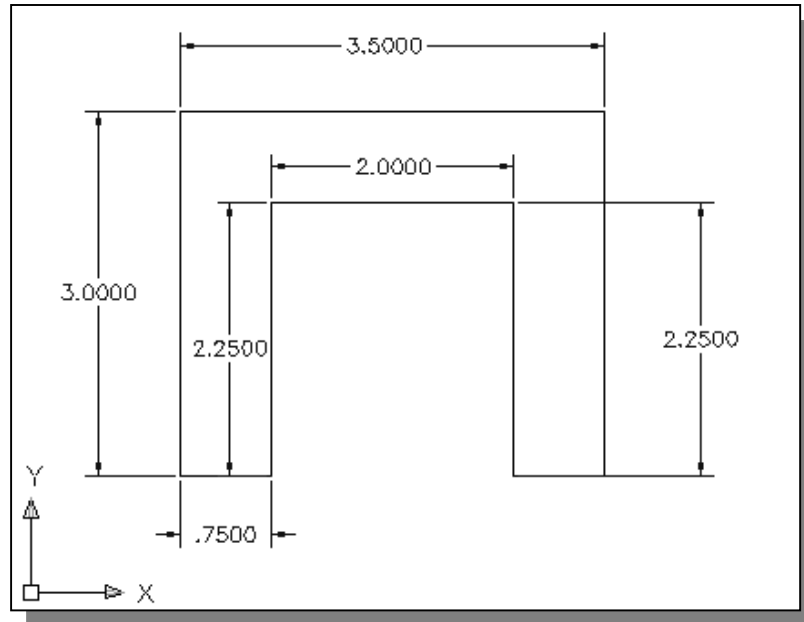


## Modifying the dimensions of the sketch



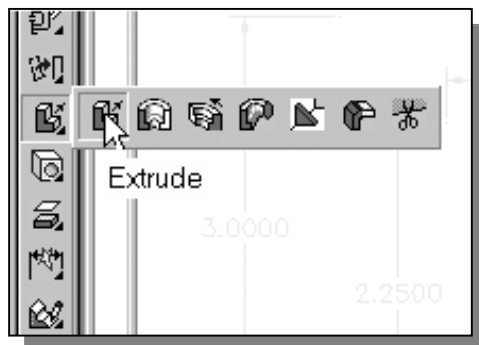
1. In the *2D Constraints* toolbar, select the **Edit Dimension** command by left-clicking once on the icon.
  2. Pick the dimension that is below the left leg of the profile.
3. In the command prompt area, the message “*Enter Dimension Value <1.8326>*” is displayed. Enter **0.75** to set the width of the leg to 0.75 inches.
- *Mechanical Desktop* will now update the profile with the new dimension value. Note that in parametric modeling, dimensions are control variables that are used to adjust the constructed geometry.

4. Pick the width dimension that is inside of the profile (the dimension positioned below the top horizontal line of the profile.)
5. In the command prompt area, the message “*Enter Dimension Value <2.4005>*” is displayed. Enter **2.0** to set the size to 2.0 inches.
6. Repeat the above steps and adjust the dimensions as shown below.



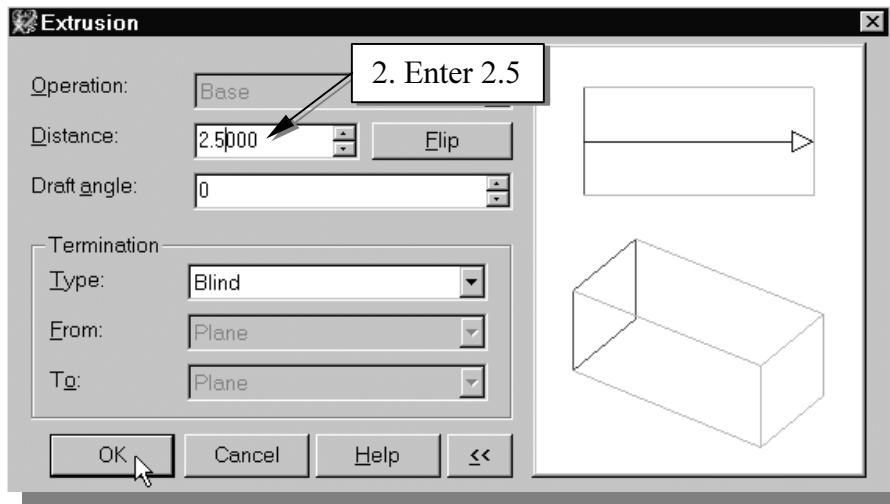
#### Step 4: Completing the Base Solid Feature

Now that the 2D sketch is completed, we will proceed to the next step: create a 3D part from the 2D profile. Extruding a 2D profile is one of the common methods that can be used to create 3D parts. We can extrude planar faces along a path. We can also specify a height value and a tapered angle. In *Mechanical Desktop*, each face has a positive side and a negative side, the current face we're working on is set as the default positive side. This positive side identifies the positive extrusion direction and it is referred to as the face's **normal**.



1. In the *Part Modeling* toolbar (the toolbar that is aligned to the left edge of the graphics window), select the **Extrude** command by releasing the left-mouse-button on the icon.

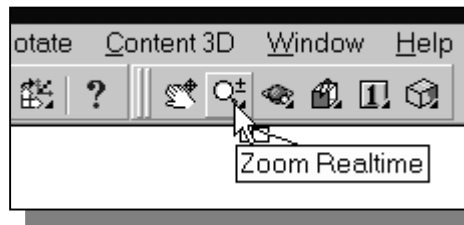
2. In the *Extrusion* popup window, enter **2.5** as the extrusion *Distance*.



3. Click on the **OK** button to proceed with creating a 3D solid model.

- Note that all dimensions disappeared from the screen. All parametric definitions are stored in the *Mechanical Desktop database* and any of the parametric definitions can be displayed and edited at anytime.

## Dynamic Viewing Functions - *Realtime Zoom and Pan*



1. Click on the **Zoom Realtime** icon in the *Standard* toolbar area.
2. Move the cursor near the center of the graphics window.
3. Inside the graphics window, **push and hold down the left-mouse-button**, then move upward to enlarge the current display scale factor.

4. Press the [**Esc**] key once to exit the *Zoom* command.

- The *Pan* command enables us to move the view to a different position. This function acts as if you are using a video camera.



5. Click on the **Pan Realtime** icon in the *Standard* toolbar area. The icon is the picture of a hand with four arrows.
6. On your own, reposition the sketch near the center of the screen.

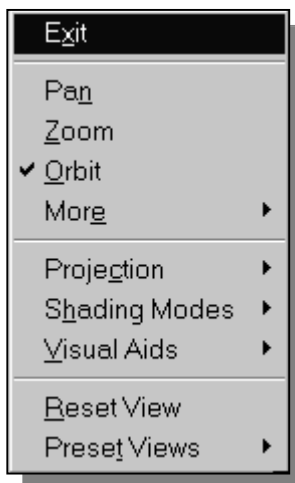
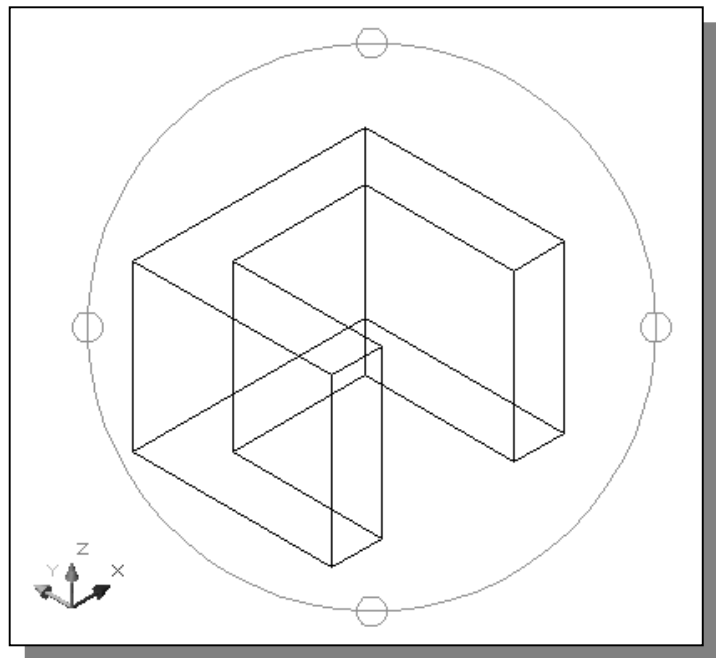
## Dynamic Rotation of the 3D block - 3D orbit



1. Click on the **3D Orbit** icon in the *Standard* toolbar.

❖ The 3D-orbit view displays an *arcball*, which is a circle divided into four quadrants by smaller circles. *3D-Orbit* enables us to manipulate the view of 3D objects by clicking and dragging with the left-mouse-button.

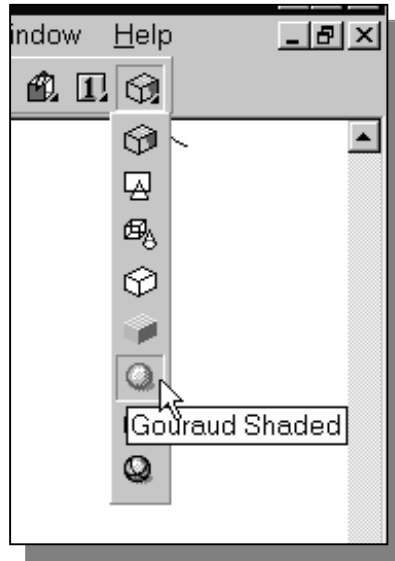
2. Inside the *arcball*, press down the left-mouse-button and drag it up and down to rotate about the screen's X-axis.
3. Move the cursor to different locations on the screen, such as outside the *arcball* or on one of the four small circles, and experiment with the real-time dynamic rotation feature of the *3D-Orbit* command.
4. On your own, rotate the 3D part so that it appears similar to the figure shown.



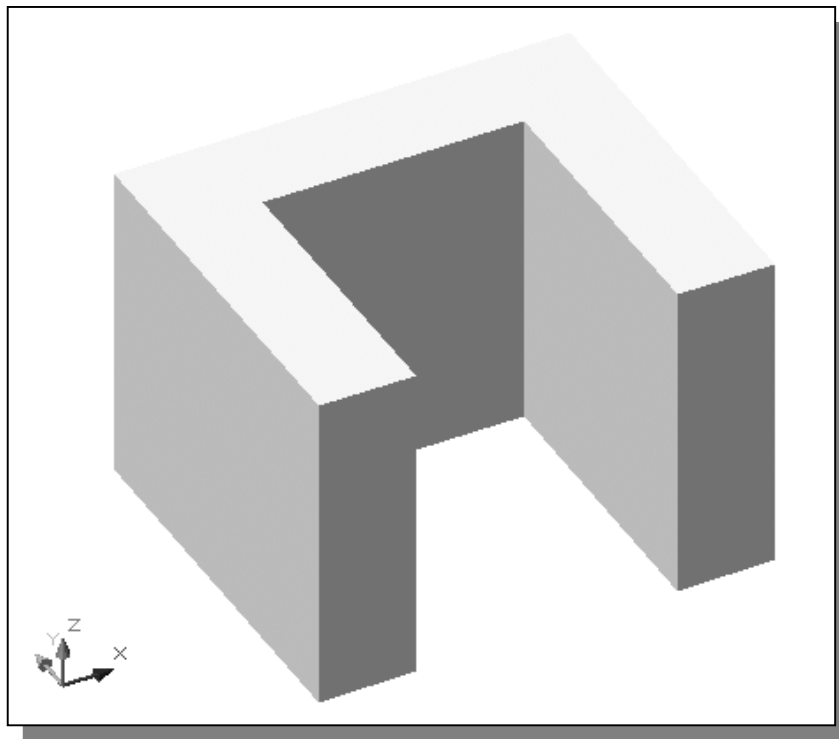
5. Inside the graphics window, click once with the right-mouse-button to display the option menu. Select **Enter** in the popup menu to end the *3D Orbit* command.



## Shaded SOLID

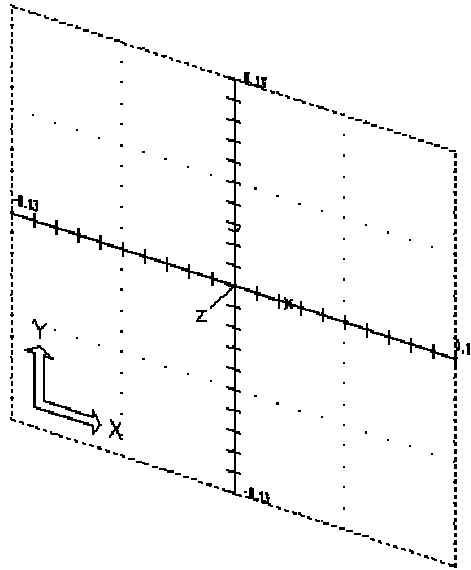


1. Move the cursor to the last icon in the *Standard* toolbar. Press and hold down the left-mouse-button on the icon to display a list of commands.
2. Select the **Gouraud Shaded** icon. This method of shading calculation produces relatively smooth three-dimensional shaded images.
  - On your own, experiment and examine the effects of the other shading options.



3. Click on the **Toggle Shading/Wireframe** icon to reset the display to wireframe.
4. On your own, rotate the 3D part so that it appears similar to the figure shown.

## Sketch plane – It is an XY CRT, but an XYZ World



Design modeling software is becoming more powerful and user friendly, yet the system still does only what the user tells it to do. When using a geometric modeler, we therefore need to have a good understanding of what the inherent limitations are. We should also have a good understanding of what we want to do and what to expect, as the results are based on what is available.

In most 3D geometric modelers, 3D objects are located and defined in what is usually called **world space** or **global space**. Although a number of different coordinate systems can be used to create and manipulate objects in a 3D modeling system, the objects are typically defined and stored using the world space. The *world space* is usually a **3D Cartesian coordinate system** that the user cannot change or manipulate.

In most engineering designs, models can be very complex, and it would be tedious and confusing if only the world coordinate system were available. Practical 3D modeling systems allow the user to define **Local Coordinate Systems (LCS)** or **User Coordinate Systems (UCS)** relative to the world coordinate system. Once a local coordinate system is defined, we can then create geometry in terms of this more convenient system.

Although objects are created and stored in 3D space coordinates, most of the geometry entities can be referenced using 2D Cartesian coordinate systems. Typical input devices such as a mouse or digitizers are two-dimensional by nature; the movement of the input device is interpreted by the system in a planar sense. The same limitation is true of common output devices, such as CRT displays and plotters. The modeling software performs a series of three-dimensional to two-dimensional transformations to correctly project 3D objects onto a 2D picture plane.

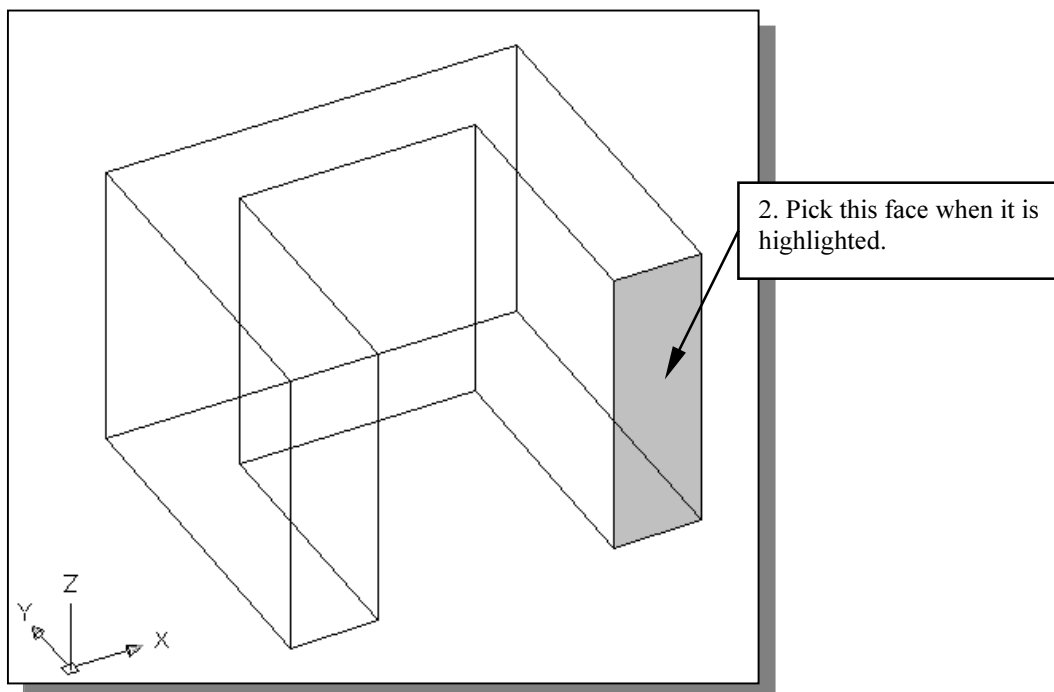
The *Mechanical Desktop's sketch plane* is a special construction tool that enables the planar nature of the 2D input devices to be directly mapped into the 3D coordinate system. The *sketch plane* is a local coordinate system that can be aligned to the world coordinate system, an existing face of a part, or a reference plane. By default, the *sketch plane* is aligned to the world coordinate system.

Think of a sketch plane as the surface on which we can sketch the 2D profiles of the parts. It is similar to a piece of paper, a white board, or a chalkboard that can be attached to any planar surface. The first profile we create is usually drawn on the default sketch plane, which is in the current coordinate system. Subsequent profiles can then be drawn on sketch planes that are defined on **planar faces of a part, work planes attached to part geometry, or sketch planes attached to the coordinate systems** (such as the World XY, XZ, and YZ sketch planes). The model we have created so far used the default settings where the sketch plane is aligned to the XY plane of the world coordinate system.

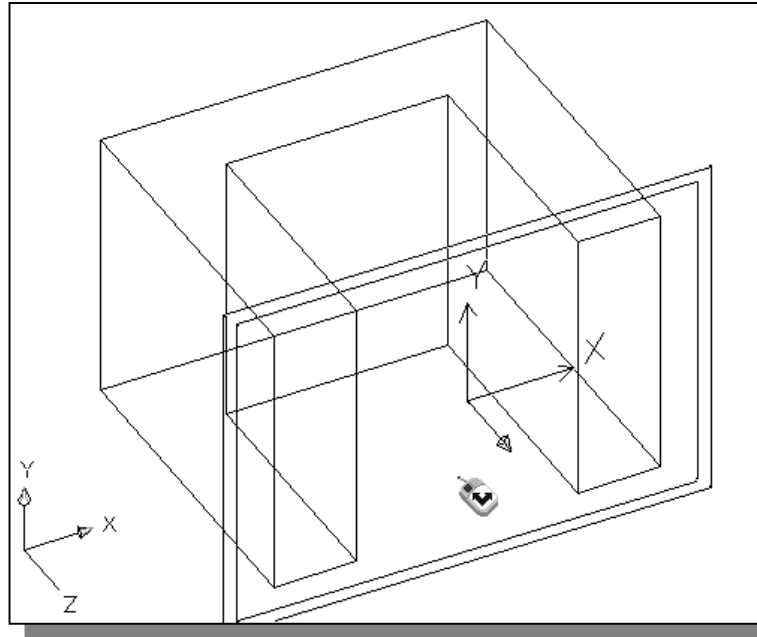


1. In the *Part Modeling* toolbar (the toolbar that is aligned to the left-edge of the graphics window), select the **New Sketch Plane** command by left-clicking once on the icon.

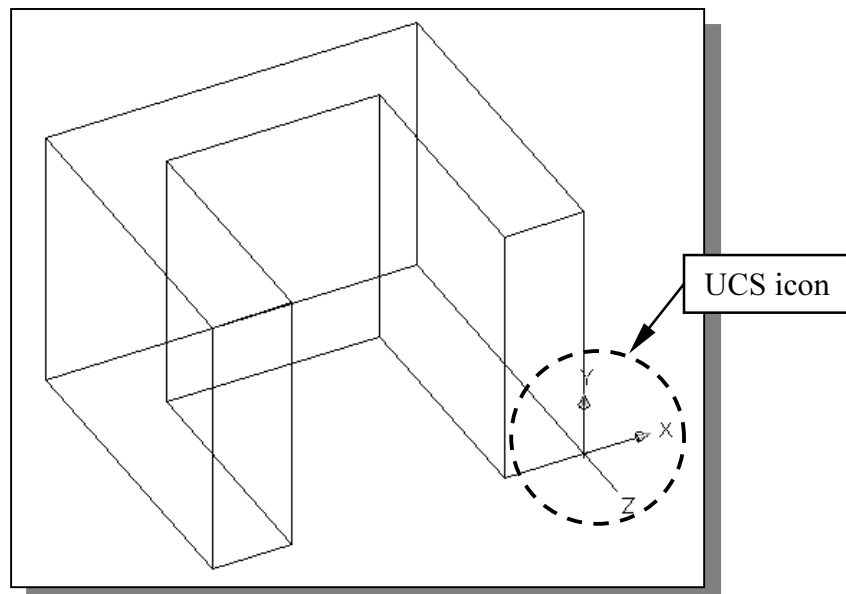
2. Move the graphics cursor on the 3D part and notice that *Mechanical Desktop* will automatically highlight feasible planes and surfaces as the cursor is on top of the different surfaces. Pick the front face of the right-leg of the 3D object as shown.



3. Confirm the selection of the front plane by a single right-mouse-click. (Left-mouse-click to choose other possible selections.)
4. In the command prompt area, the message “*Select edge to align X-axis or {Z-flip/Rotate} <Accept>*” is displayed. Use the left-mouse-button to rotate the alignment of the coordinate axes. Accept the orientation of the axes, by right-mouse-clicking once, when the display appears as shown below.

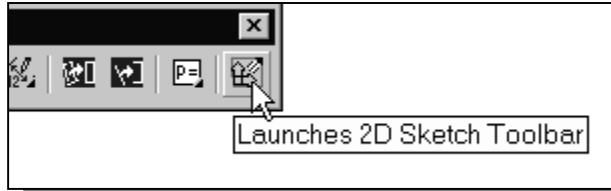


- Note that the sketch plane is not visible, *Mechanical Desktop* repositions the User-Coordinate-System (UCS), and records its location with respect to the part on which it was created. The orientation of the UCS icon is the only way we can easily see where a sketch plane is located.

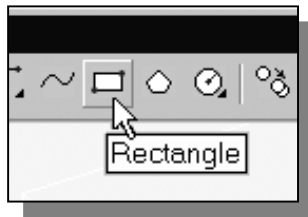


## Step 5: Adding additional features

- Next, we will create and profile a rectangle, which will be used to create an extension to the right side of the current 3D object.

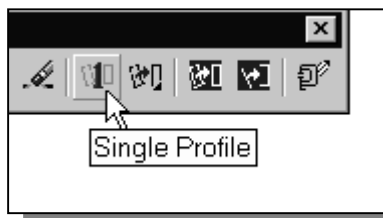
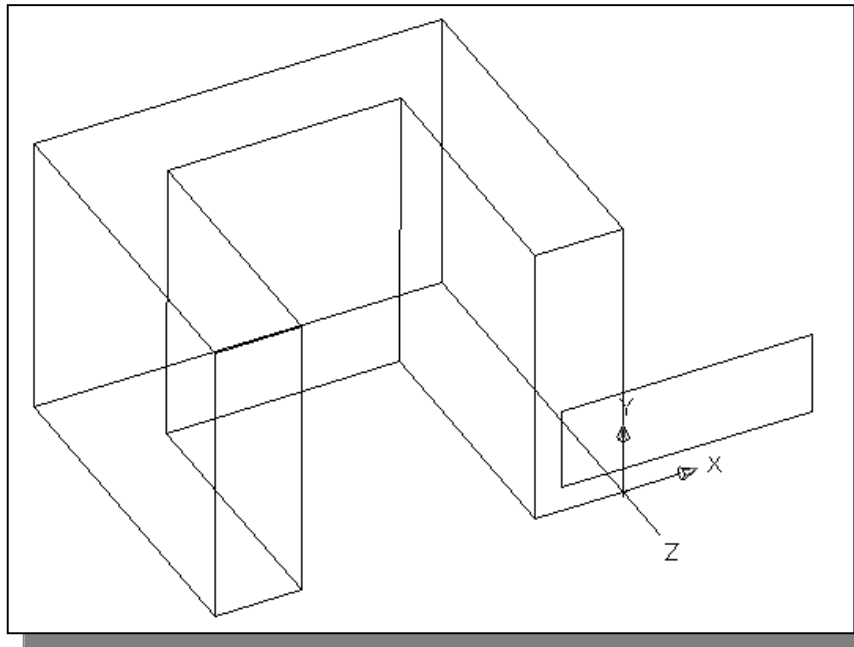


1. Select the last icon in the *2D Constraints* toolbar. This icon allows us quickly switch to the *2D Sketching* toolbar.



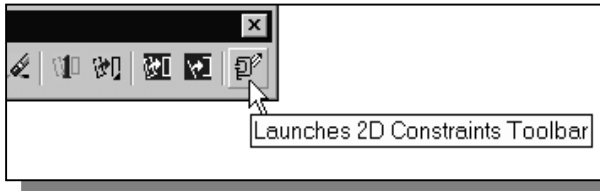
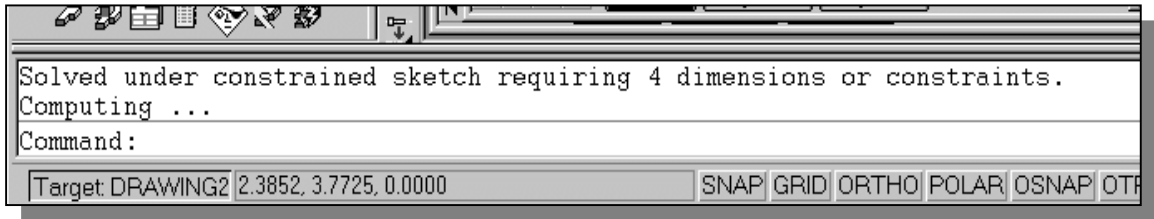
2. Select the **Rectangle** command by clicking once with the left-mouse-button on the icon.

3. Create a rectangle by picking two locations on the sketch plane, as shown below. Do not be overly concerned with the size of the rectangle; we are making a rough sketch.

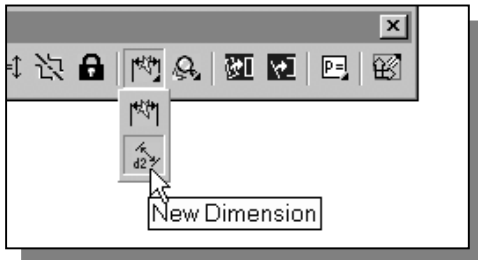


4. Select **Single Profile** by left-clicking once on the icon in the *2D Sketching* toolbar. This command can be used to quickly convert a single curve to a parametric profile.

- The circle is converted to a profile, which still require four dimensions or constraints as indicated in the command prompt area. On your own, identify a set of four dimensions that can be applied to the profile so that the geometry is fully defined.

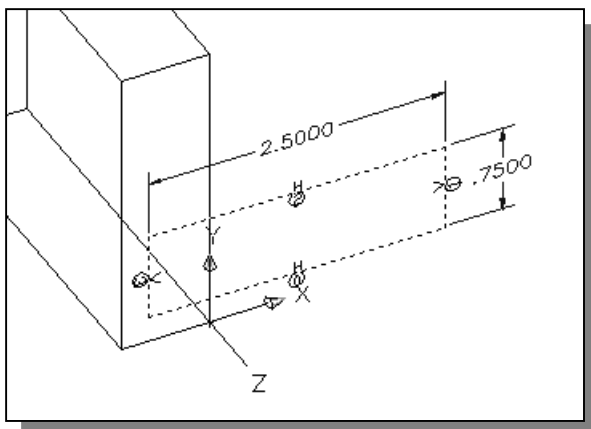
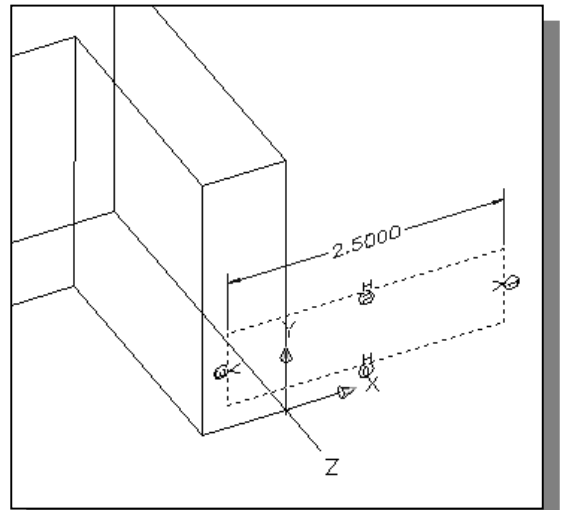


5. Left-click once on the *Launches 2D Constraints Toolbar* icon to display the *2D Constraints* toolbar.

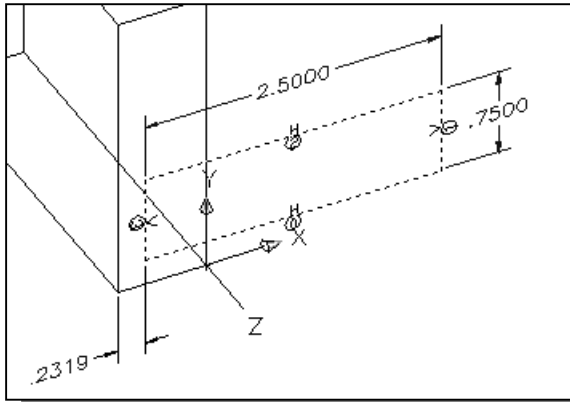


6. In the *2D Constraints* toolbar, select the **New Dimension** command by left-clicking once on the icon.

7. Pick the top edge of the rectangle as the first object to dimension.
8. Pick a location that is above the selected edge to place the dimension.
9. Enter **2.5** in the command prompt area to adjust the length of the rectangle to 2.5 inches.



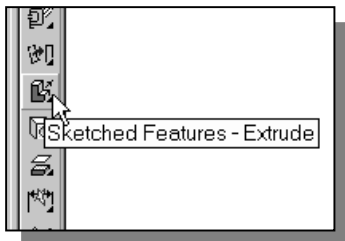
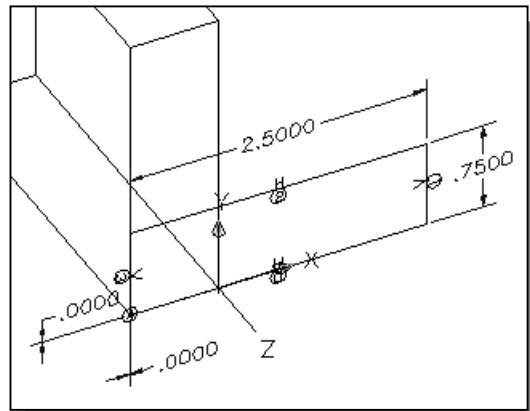
10. Repeat the above step and create a height dimension (**0.75** inches) of the rectangle as shown.
  - ❖ What are the other two dimensions needed to fully describe the rectangle in relation to the solid model.



11. Next create a location-dimension by first clicking on the left edge of the profile and then clicking on the left edge of the right-leg of the part to create a dimension as shown.

12. Enter **0.0** in the command prompt area to align the two selected edges.

13. On your own, repeat the above step and align the bottom edge of the profile to the bottom edge of the 3D model as shown.

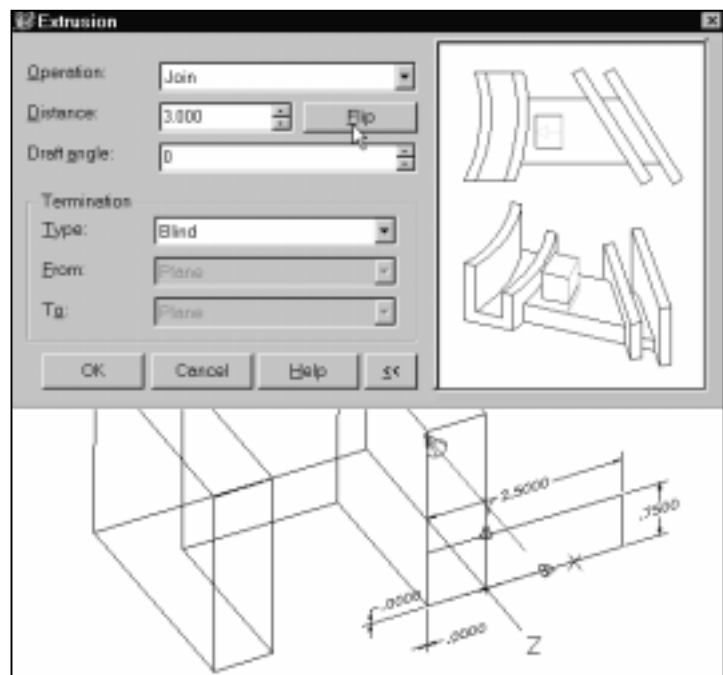


14. In the *Part Modeling* toolbar (the toolbar that is aligned to the left edge of the graphics window), select the **Extrude** command by releasing the left-mouse-button on the icon.

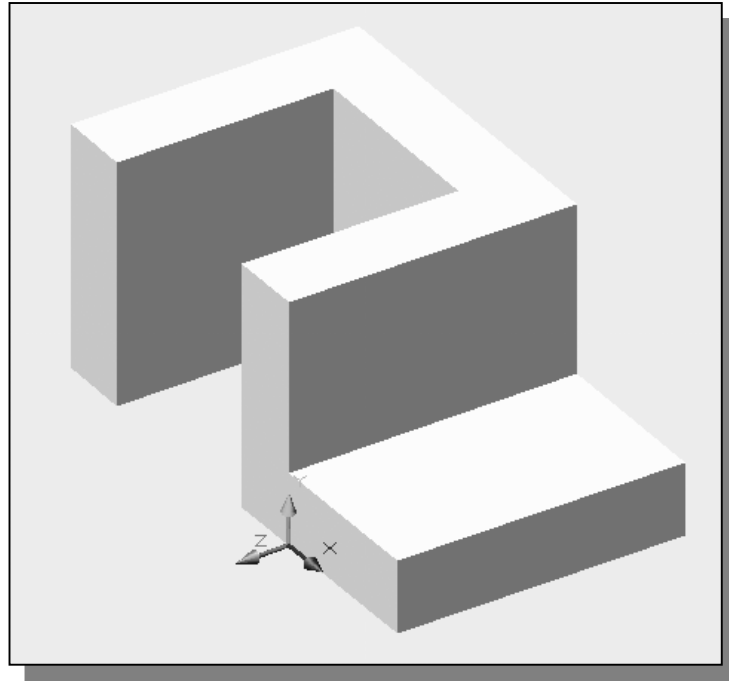
15. In the *Extrusion* popup window, select **Join** as the extrusion *Operation* method. Enter **3.0** as the extrusion *Distance*.

16. Click on the **Flip** button to set the extrusion direction as shown. (Move the pop-up window by dragging the title area of the window)

17. Click on the **OK** button to proceed with creating the feature.



- ❖ Note the rectangular feature is added to the base solid, and all the surfaces and edges are merged and re-constructed to form the 3D solid model.



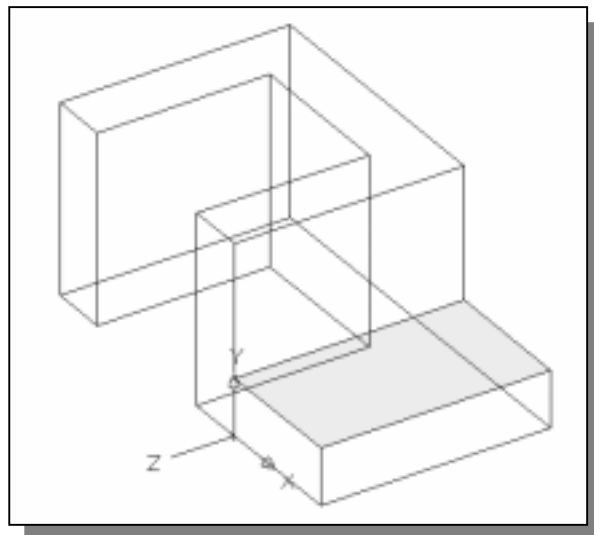
### Step 6: Adding a cut feature and Complete the design

- Next, we will create and profile a circle, which will be used to create a cut feature and complete the 3D model.

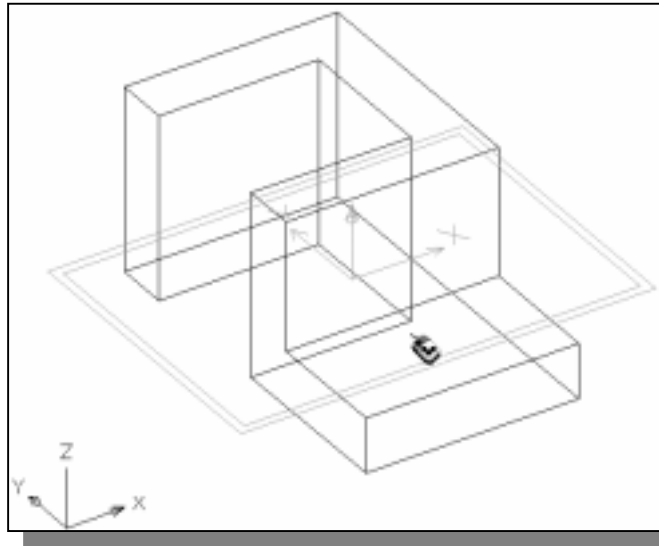


1. In the *Part Modeling* toolbar (the toolbar that is aligned to the left-edge of the graphics window), select the *New Sketch Plane* command by left-clicking once on the icon.

2. Pick the horizontal face of the last feature of the 3D object as shown.



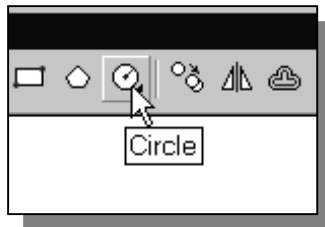




3. Confirm the selection of the horizontal plane by a single right-mouse-click. (Left-mouse-click to choose other possible selections.)
4. In the command prompt area, the message “*Select edge to align X-axis or {Z-flip/Rotate} <Accept>*” is displayed. Use the left-mouse-button to rotate the alignment of the coordinate axes. Accept the orientation of the axes, by right-mouse-clicking once, when the display appears as shown.

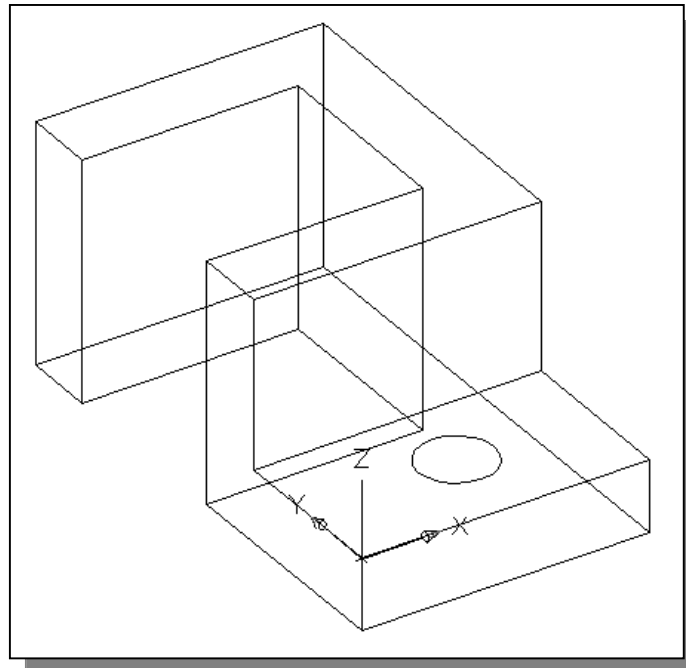


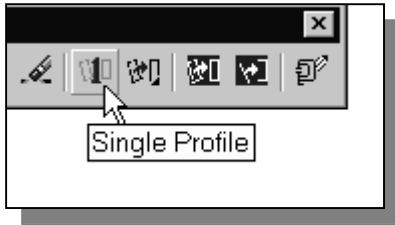
5. Select the last icon in the *2D Constraints* toolbar. This icon allows us quickly switch to the *2D Sketching* toolbar.



6. Select the **Circle** command by clicking once with the left-mouse-button on the **Circle** icon.

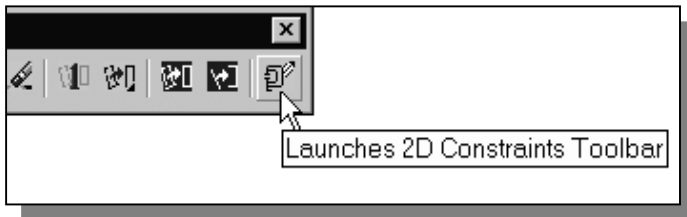
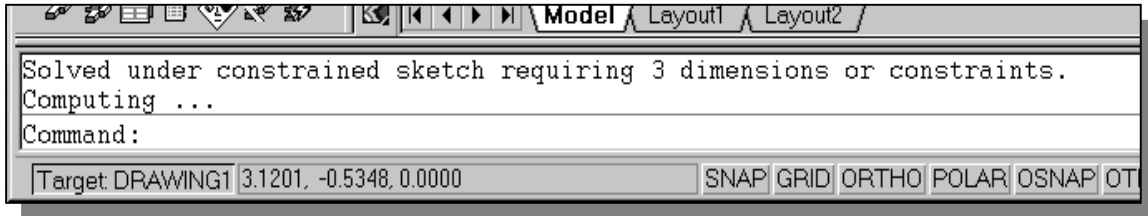
7. Create a circle by picking two locations inside the rectangular surface of the 3D part, as shown below. Do not be overly concerned with the size and location of the circle; we are making a rough sketch.



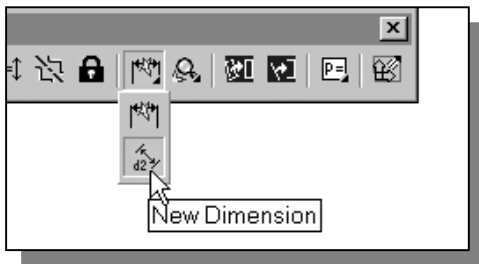


8. Select **Single Profile** by left-clicking once on the icon in the *2D Sketching* toolbar. This command can be used to quickly convert a single curve to a parametric profile.

- The circle is converted to a profile, which still require three dimensions or constraints as indicated in the command prompt area.

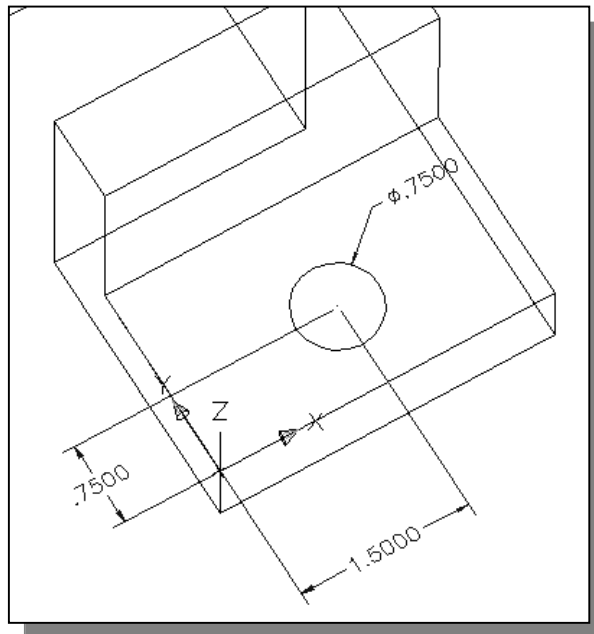


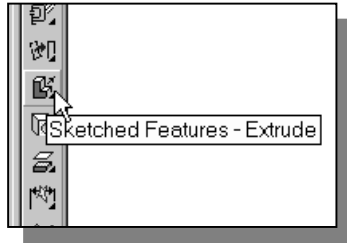
9. Left-click once on the **Launches 2D Constraints Toolbar** icon to display the *2D Constraints* toolbar.



10. In the *2D Constraints* toolbar, select the **New Dimension** command by left-clicking once on the icon.

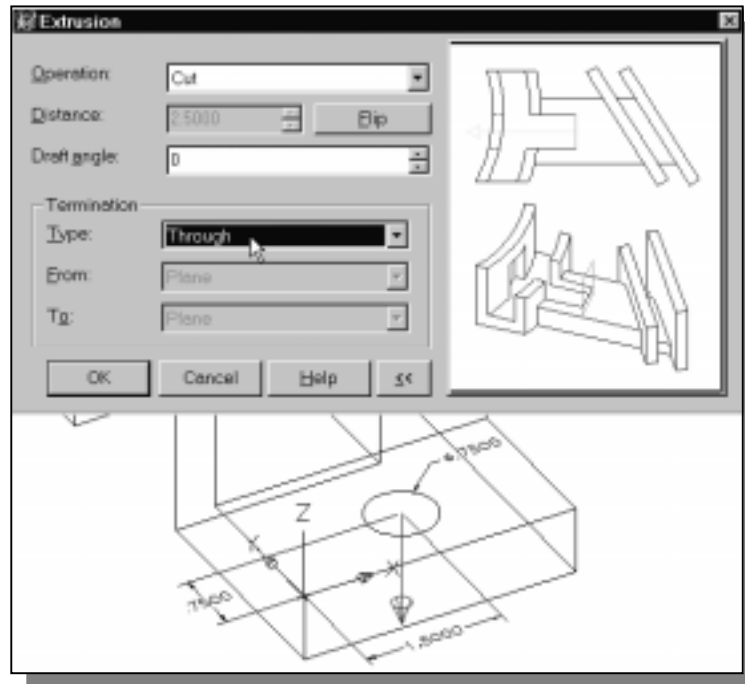
- 11. On your own, create the three dimensions as shown.
- ❖ The three dimensions provide the complete definition of the size and location of the circle. The profile is fully defined.



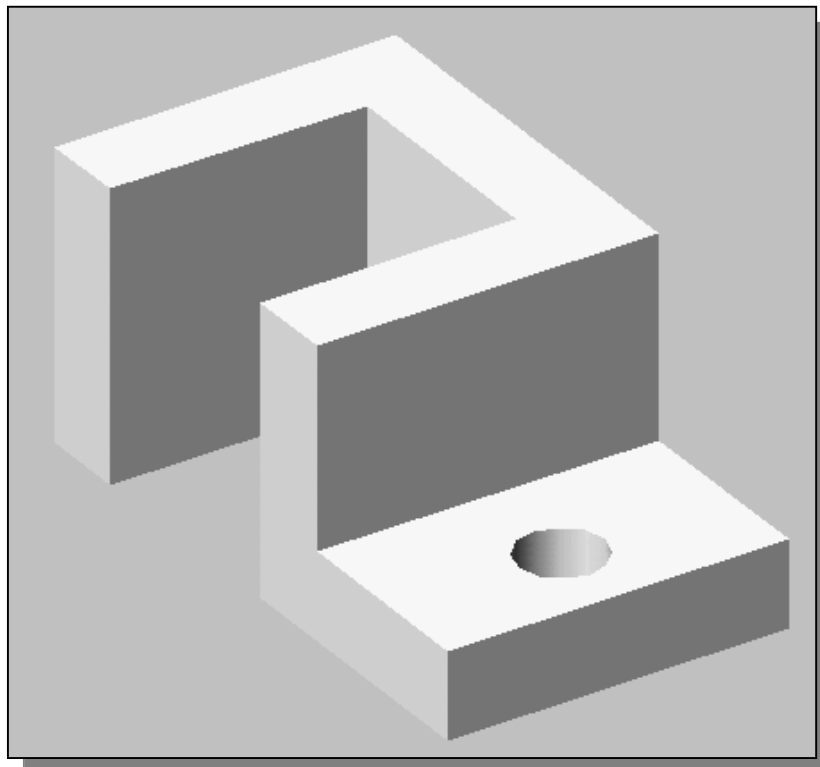


12. In the *Part Modeling* toolbar (the toolbar that is aligned to the left edge of the graphics window), select the **Extrude** command by releasing the left-mouse-button on the icon.

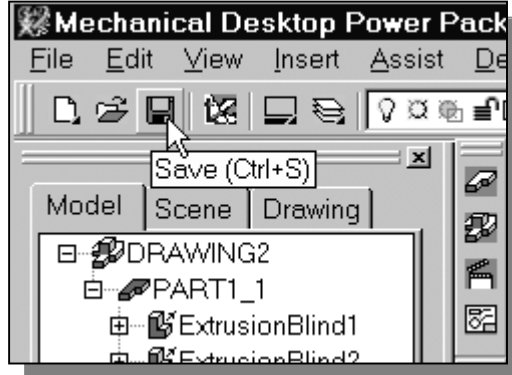
13. In the *Extrusion* popup window, set the *Operation* to **Cut** and select **Through** as the extrusion *Termination* method. Confirm that the arrowhead points downward as shown below.



14. Click on the **OK** button to proceed with creating the cut feature.



## Save the Part and Exit

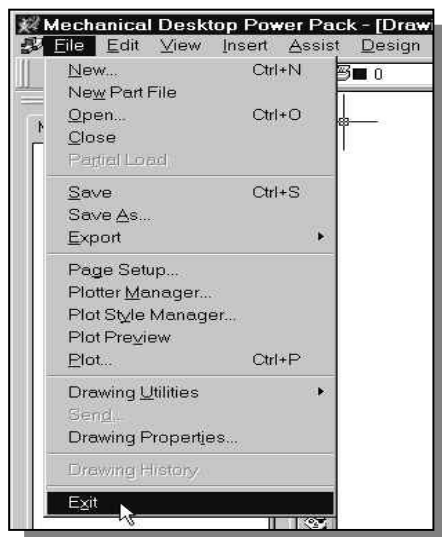


1. Select **Save** in the *Standard* toolbar. We can also use the “**Ctrl-S**” combination (press down the [Ctrl] key and hit the [S] key once) to save the part.

2. In the popup window, select the folder to place the completed model and enter **Adjuster** as the name of the file.
3. Confirm the file type is set to *Mechanical Desktop 5 Drawing* file as shown.
4. Click on the **SAVE** button to save the file.



- ❖ It is a good habit to save your model periodically, just in case something might go wrong while you are working on it. In general, you should save your work onto the disk at an interval of every 15 to 20 minutes. You should also save before you make any major modifications to the model.



5. Now you can leave *Mechanical Desktop*. Use the left-mouse-button and click on **File** at the top of the *Mechanical Desktop* screen window, then choose **Exit** from the pull-down menu or type **QUIT** at the command prompt area.

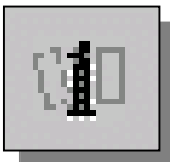
**Questions:**

1. What is the first thing we should set up in *Mechanical Desktop* when creating a new model?
2. Describe the general *parametric modeling* procedure.
3. What is the main difference between a *rough sketch* and a *profile*?
4. List two of the geometric constraint symbols used by the *Mechanical Desktop*.
5. What was the first feature we created in this lesson?
6. Describe the steps required to define the orientation of the sketching plane?
7. Identify the following commands:

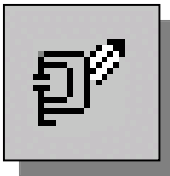
(a)



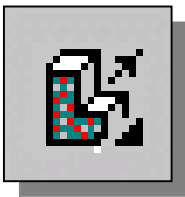
(b)



(c)

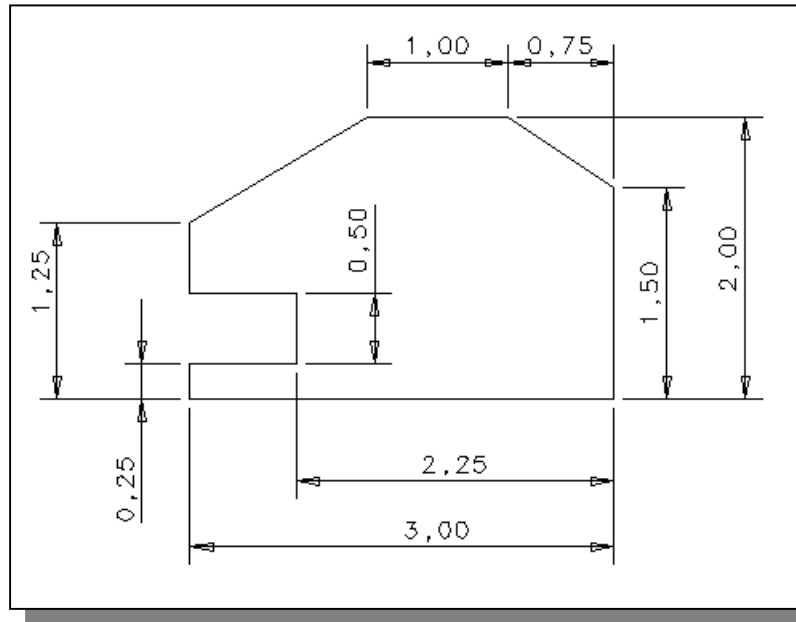


(d)

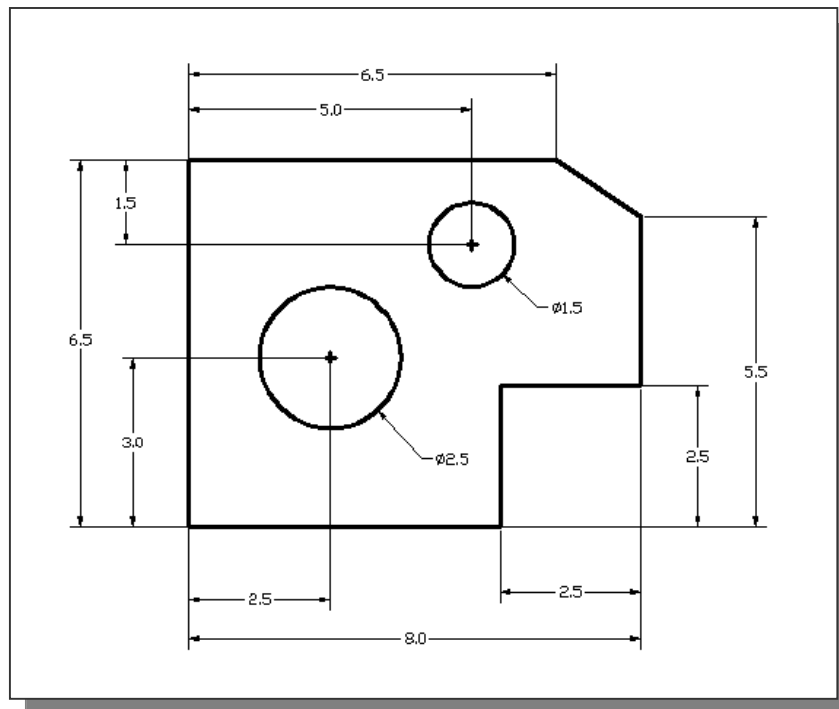


**Exercises:** (All dimensions are in inches.)

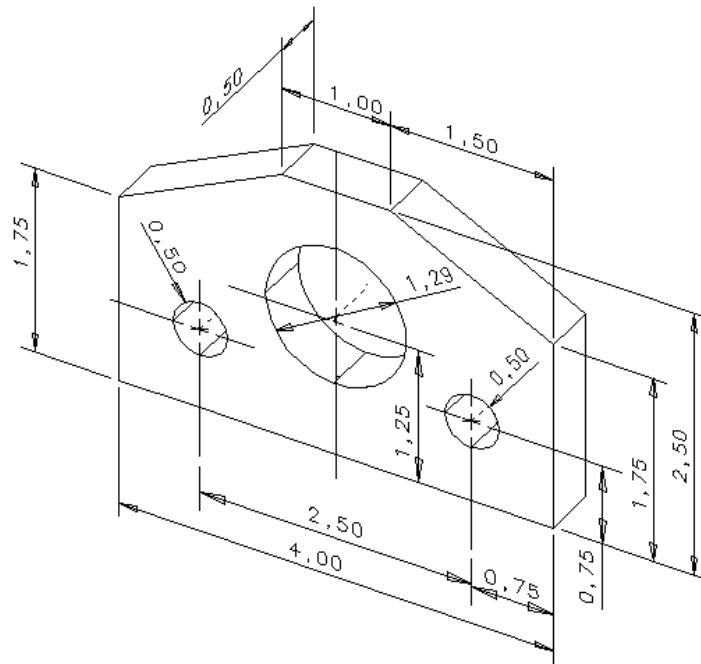
1. Plate Thickness: 0.25



2. Plate Thickness: 0.5



3.



4.

