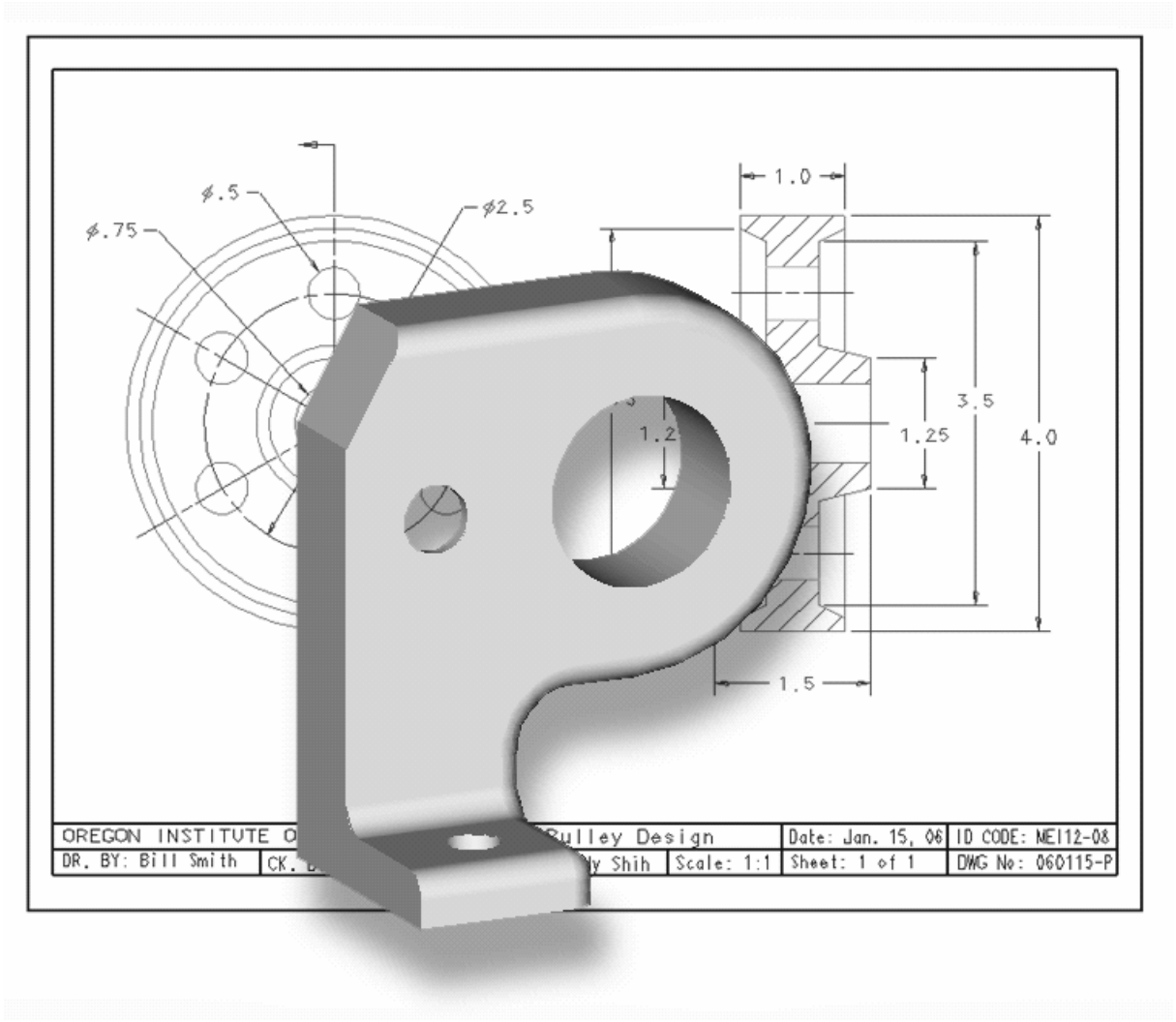
Parametric Modeling

With

I-DEAS 12®



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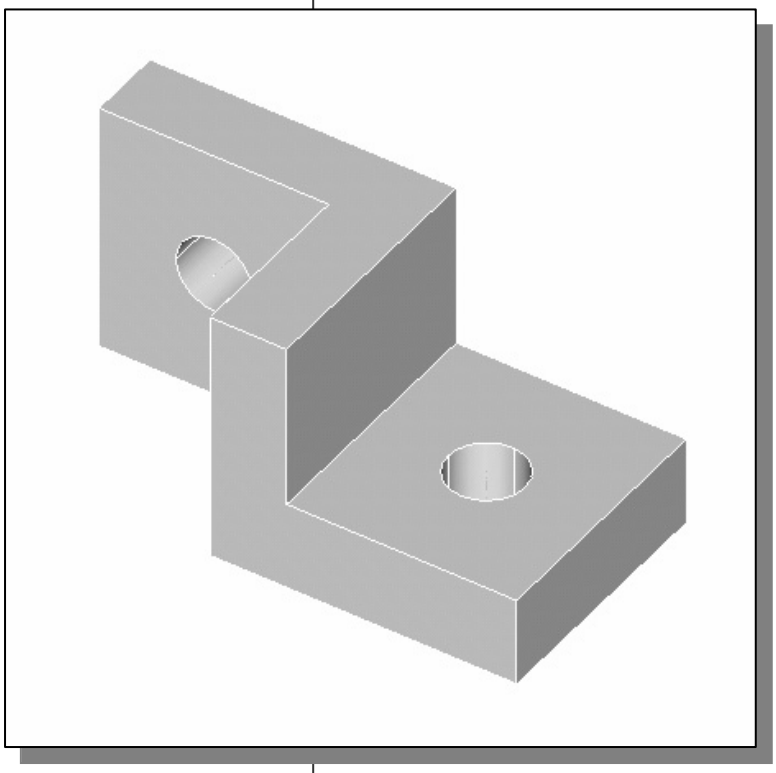
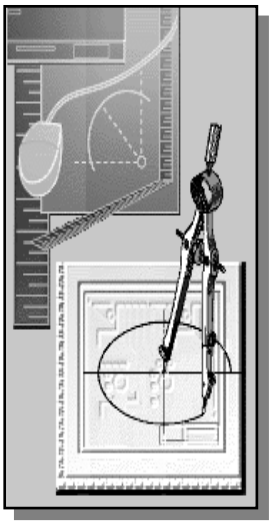
Chapter 2
Parametric Modeling Fundamentals





Learning Objectives

- Understand the Parametric Part Modeling process
- Understand the basic functions of the Dynamic Navigator.
- Create Rough Sketches
- Understand the "Shape before size" approach.
- Use the Dynamic Viewing commands.
- Use the Basic Modify commands.

Introduction

The **feature-based parametric modeling** technique enables the designer to incorporate the original **design intent** into construction of the model. The word ***parametric*** means that geometric definitions of the design, such as dimensions, can be varied at any time in the design process. Parametric modeling is accomplished by identifying and creating the key features of the design with the aid of computer software. The design variables, described in sketches and described as parametric relations, can then be used to quickly modify/update the design.

In *I-DEAS*, 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. **Apply/delete/modify constraints and dimensions to the two-dimensional sketch.**

3. **Extrude, revolve, or sweep the parametric two-dimensional sketch to create the first solid feature, the base feature, of the design.**

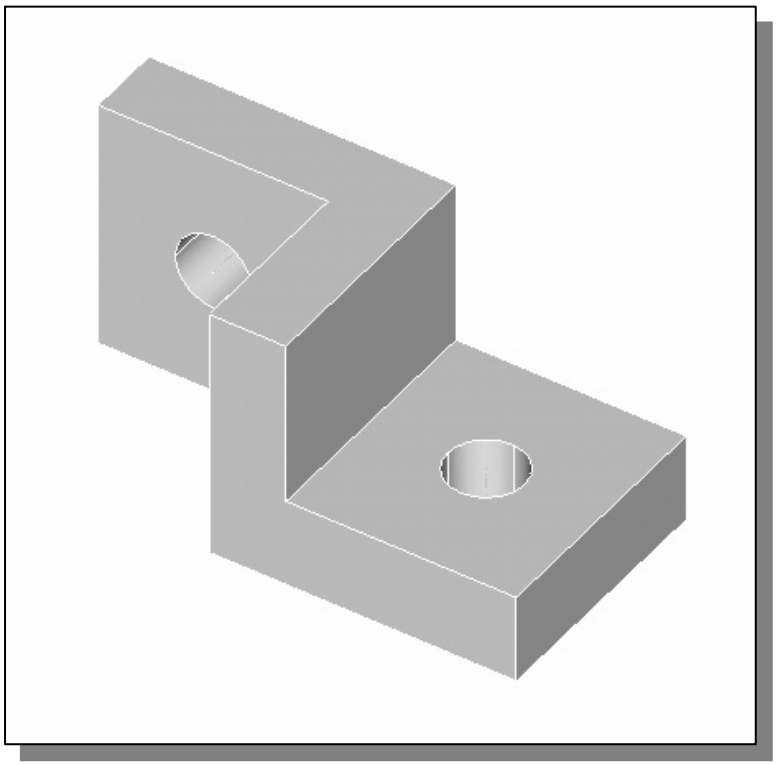
4. **Add additional parametric features by identifying feature relations and complete the design.**

5. **Perform analyses on the computer model and refine the design as needed.**

6. **Create the desired drawing views to document the design.**

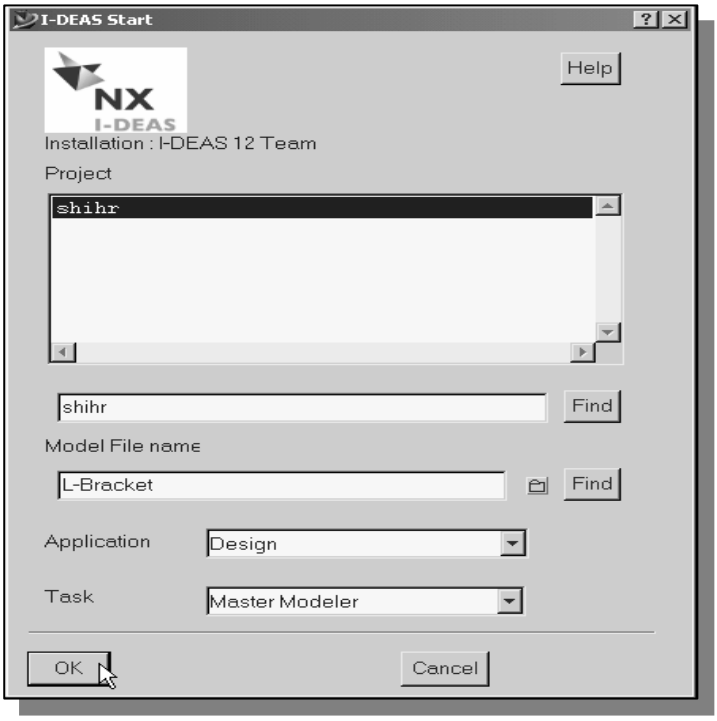
The approach of creating two-dimensional sketches of the three-dimensional features is an effective way to construct solid models. Many designs are in fact the same shape in one direction. Computer input and output devices we use today are largely two-dimensional in nature, which makes this modeling 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. *I-DEAS* 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 *I-DEAS* user interface. The display viewing functions and the basic two-dimensional sketching tools are also demonstrated.

The *L-Bracket* Design



Starting *I-DEAS*

1. Select the **I-DEAS** icon or type "*ideas*" at your system prompt to start *I-DEAS*. The *I-DEAS Start* window will appear on the screen.



2. Fill in and select the items as shown below:

Project Name: **(Your account name)**
Model File Name: **L-Bracket**
Application: **Design**
Task: **Master Modeler**

3. After you click **OK**, two *warning windows* will appear to tell you that a new model file will be created. Click **OK** on both windows as they come up.

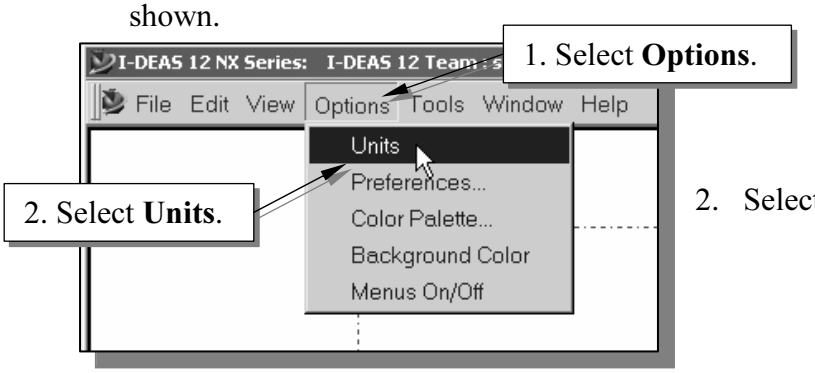
I-DEAS Warning
! New Model File will be created
OK

4. Next, *I-DEAS* will display the main screen layout, which includes the *graphics window*, the *prompt window*, the *list window* and the *icon panel*. A line of *quick help* text appears at the bottom of the graphics window as you move the mouse cursor over the icons.

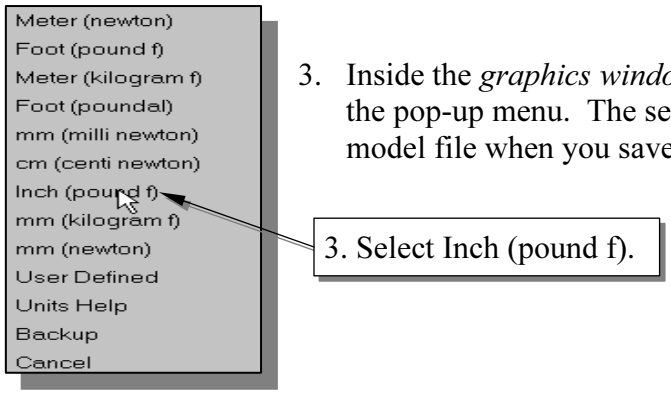
Units Setup

❖ When starting a new model, the first thing we should do is determine the set of units we would like to use. *I-DEAS* displays the default set of units in the *list window*.

1. Use the left-mouse-button and select the **Options** menu in the icon panel as shown.



2. Select the **Units** option.



3. Inside the *graphics window*, pick **Inch (pound f)** from the pop-up menu. The set of units is stored with the model file when you save.

Shape Before Size

Quite often during the early design stage, the shape of a design may not have any precise dimensions. Most conventional CAD systems require the user to input precise lengths and locations of all geometric entities defining the design, which are not available during the early design stage. With *parametric modeling*, we can use the computer to elaborate and formulate the design idea further during the initial design stage. With *I-DEAS*, 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. *I-DEAS* provides us with many tools to assist us in finalizing sketches. For example, geometric entities such as horizontal and vertical lines are set automatically. However, if the rough sketches are poor, it will require much more work to generate the desired parametric sketches. Here are some general guidelines for creating sketches in *I-DEAS*:

- **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 geometric 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, *I-DEAS* might assume the intended angle to be a 90 degree angle.

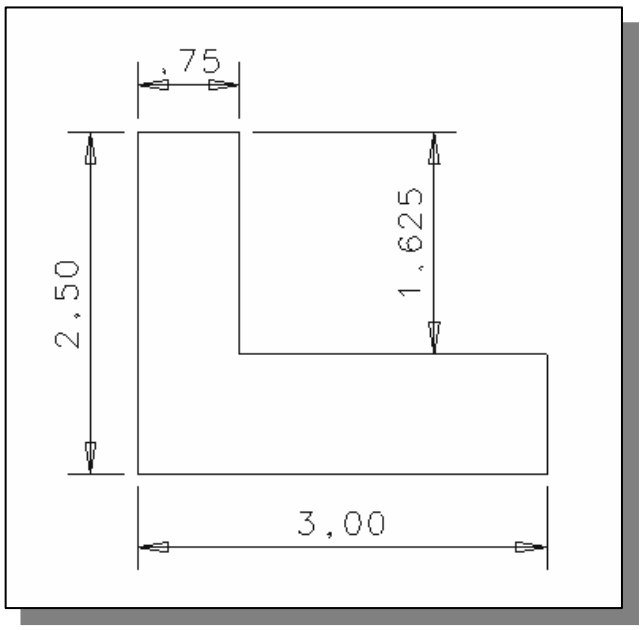
- **Draw the geometry so that it does not overlap.** *Self-intersecting* geometric shapes and identical geometry placed at the same location are not allowed.

- **The sketched geometric entities should form a closed region.** To create a solid feature such as an extruded solid, a closed region is required so that the extruded solid forms a 3D volume.

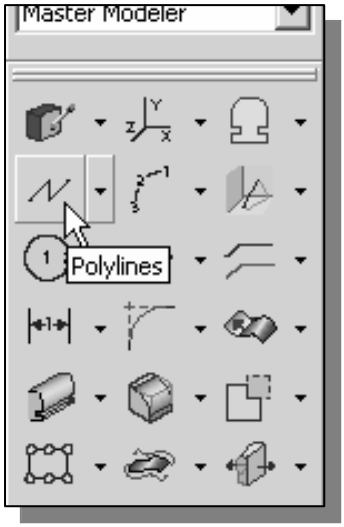
➢ **Note:** The concepts and principles involved in *parametric modeling* are very different, and sometimes they are totally opposite, those of conventional two-dimensional computer aided drafting systems. In order to understand and fully utilize *I-DEAS's* functionality, it will be helpful to take a *Zen* approach to learning the topics presented in this text: **Temporarily forget your knowledge and experiences of using conventional 2D Computer Aided Drafting systems.**

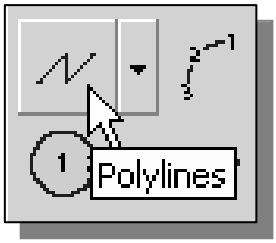
Step 1: Creating a Rough Sketch

❖ In this lesson we will begin by building a 2D sketch, as shown in the figure below.



I-DEAS provides many powerful tools for sketching 2D shapes. In the previous generation CAD programs, exact dimensional values were needed during construction, and adjustments to dimensional values were quite difficult once the model is built. In *I-DEAS*, we can now treat the sketch as if it is being done on a piece of napkin, and it is the general shape of the design that we are more interested in defining. The *I-DEAS* part model contains more than just the final geometry; it also contains the **design intent** that governs what will happen when geometry changes. The design philosophy of "shape before size" is implemented through the use of ***I-DEAS'* Variational Geometry**. This allows the designer to construct solid models in a higher level and leave all the geometric details to *I-DEAS*. We will first create a rough sketch, by using some of the visual aids available, and then update the design through the associated control parameters.

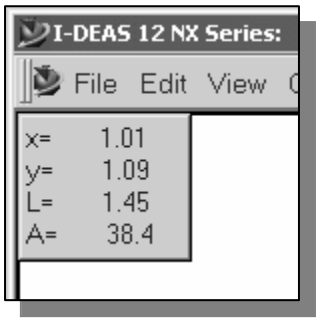




1. Pick **Polylines** in the icon panel. (The icon is located in the second row of the task specific icon panel. If the icon is not on top of the stack, press and hold down the left-mouse-button on the displayed icon to display all the choices. Select the desired icon by clicking with the left-mouse-button when the icon is highlighted.)

Graphics Cursors

- Notice the cursor changes from an arrow to a crosshair when graphical input is expected. Look in the prompt window for a description of what you are to choose. The cursor will change to a *double crosshair* when there is a possibly ambiguous choice. When the *double crosshair* appears, you can press the middle-mouse-button to accept the highlighted pick or choose a different item.

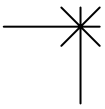


2. The message "*Locate start*" is displayed in the *prompt window*. Left-click a starting point of the shape, roughly at the center of the graphics window; it could be inside or outside of the displayed grids. In *I-DEAS*, the sketch plane actually extends into infinity. As you move the graphics cursor, you will see a digital readout in the upper left corner of the graphics window. The readout gives you the cursor location, the line length, and the angle of the line measured from horizontal. Move the cursor around and you will also notice different symbols appear along the line as it occupies different positions.

Dynamic Navigator

I-DEAS provides you with visual clues as the cursor is moved across the screen; this is the *I-DEAS Dynamic Navigator*. The *Dynamic Navigator* displays different symbols to show you alignments, perpendicularities, tangencies, etc. The *Dynamic Navigator* is also used to capture the *design intent* by creating constraints where they are recognized. The *Dynamic Navigator* displays the governing geometric rules as models are built.

	**Vertical**	indicates a line is vertical
	**Horizontal**	indicates a line is horizontal
------	**Alignment**	indicates the alignment to the center point or endpoint of an entity
	**Parallel**	indicates a line is parallel to other entities
	**Perpendicular**	indicates a line is perpendicular to other entities
	**Endpoint**	indicates the cursor is at the endpoint of an entity

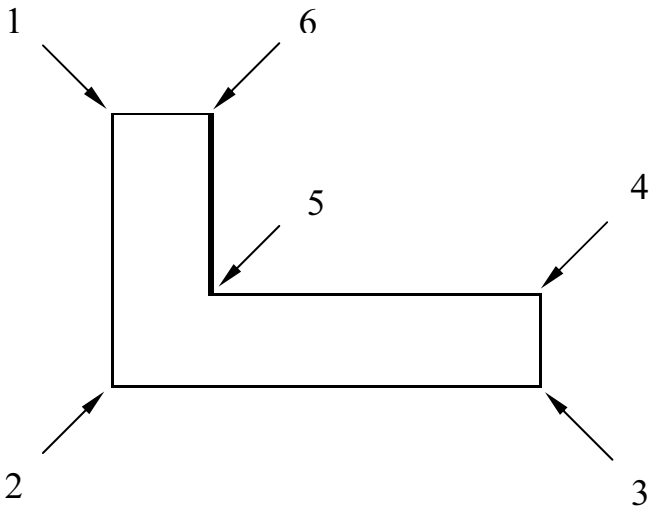


Intersection indicates the cursor is at the intersection point of two entities

Center indicates the cursor is at the centers or midpoints of entities

Tangent indicates the cursor is at tangency points to curves

3. Move the graphics cursor directly below *point 1*. Pick the second point when the *vertical constraint* is displayed and the length of the line is about 2 inches.



4. Move the graphics cursor horizontally to the right of *point 2*. The *perpendicular* symbol indicates when the line from *point 2* to *point 3* is perpendicular to the vertical line. Left-click to select the third point. Notice that dimensions are automatically created as you sketch the shape. These dimensions are also constraints, which are used to control the geometry. Different dimensions are added depending upon how the shape is sketched. Do not worry about the values not being exactly what we want. We will modify the dimensions later.

5. Move the graphics cursor directly above *point 3*. Do not place this point in alignment with the midpoint of the other vertical line. An additional constraint will be added if they are aligned. Left-click the fourth point directly above *point 3*.

6. Move the graphics cursor to the left of *point 4*. Again, watch the displayed symbol to apply the proper geometric rule that will match the design intent.

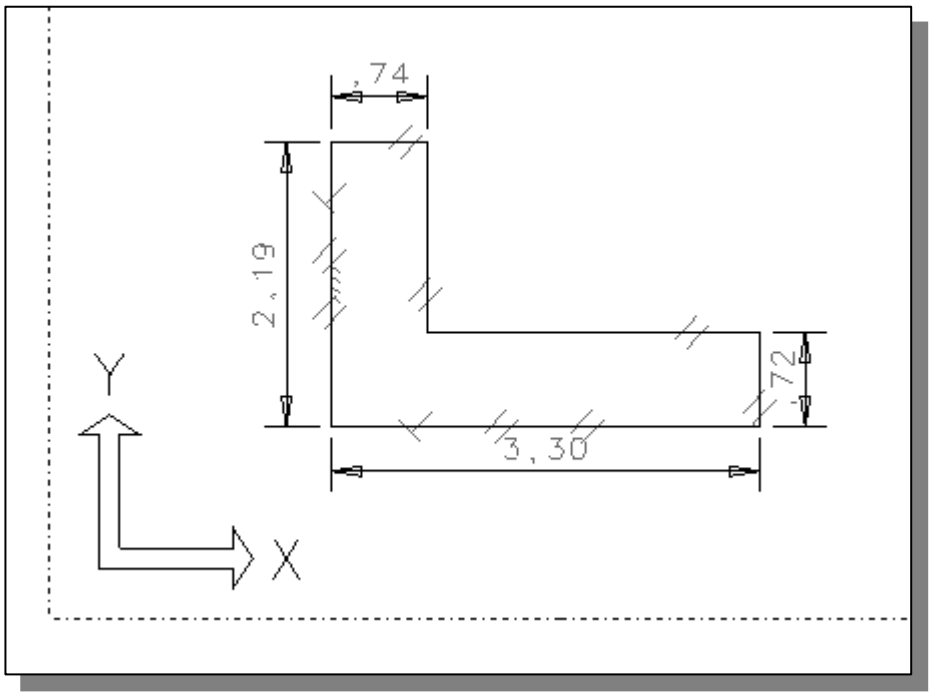
A good rule of thumb is to exaggerate the features during the initial stage of sketching. For example, if you want to construct a line that is five degrees from horizontal, it would be easier to sketch a line that is 20 to 30 degrees from horizontal. We will be able to adjust the actual angle later. Left-click once to locate the fifth point horizontally from *point 4*.

7. Move the graphics cursor directly above the last point. Watch the different symbols displayed and place the point in alignment with *point 1*. Left-click the sixth point directly above *point 5*.

8. Move the graphics cursor near the starting point of the sketch. Notice the *Dynamic Navigator* will jump to the endpoints of entities. Left-click *point 1* again to end the sketch.

9. In the prompt window, you will see the message "*Locate start*." By default, *I-DEAS* remains in the **Polylines** command and expects you to start a new sequence of lines.

10. Press the **ENTER** key or click once with the middle-mouse-button to end the **Polylines** command.



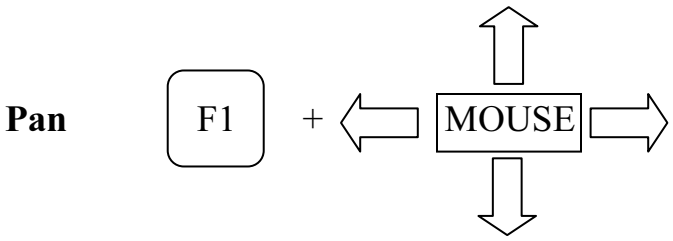
- Your sketch should appear similar to the figure above. Note that the displayed dimension values may be different on your screen. In the following sections, we will discuss the procedure to adjust the dimensions. At this point in time, our main concern is creating the proper **SHAPE** of the sketch.

Dynamic Viewing Functions

I-DEAS provides a special user interface called *Dynamic Viewing* that enables convenient viewing of the entities in the graphics window. The *Dynamic Viewing* functions are controlled with the function keys on the keyboard and the mouse.

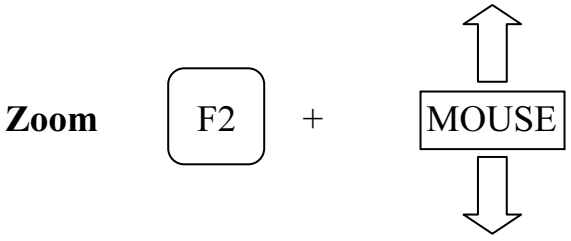
❖ Panning – F1 and the mouse

Hold the **F1** function key down, and move the mouse to pan the display. This allows you to reposition the display while maintaining the same scale factor of the display. This function acts as if you are using a video camera. You control the display by moving the mouse.



❖ Zooming – (1) F2 and the mouse

Hold the **F2** function key down, and move the mouse vertically on the screen to adjust the scale of the display. Moving upward will reduce the scale of the display, making the entities display smaller on the screen. Moving downward will magnify the scale of the display.



❖ Zooming – (2) The mouse wheel

Turning the mouse wheel can also adjust the scale of the display. Turning forward will reduce the scale of the display, making the entities display smaller on the screen. Turning backward will magnify the scale of the display.

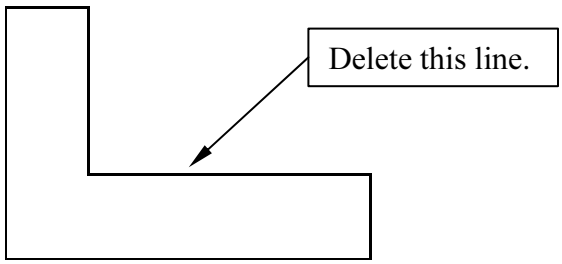
♦ On your own, experiment with the two *Dynamic Viewing* functions. Adjust the display so that your sketch is near the center of the graphics window and adjust the scale of your sketch so that it is occupies about two-thirds of the graphics window.

Basic Editing – Using the Eraser

One of the advantages of using a CAD system is the ability to remove entities without leaving any marks. We will delete one of the lines using the ***Delete*** command.

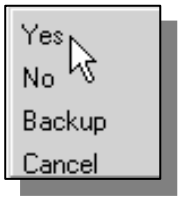
1. Pick **Delete** in the icon panel. (The icon is located in the last row of the *application icon panel*. The icon is a picture of an eraser at the end of a pencil.)

2. In the prompt window, the message "*Pick entity to delete*" appears. Pick the line as shown in the figure below.



3. The prompt window now reads "*Pick entity to delete (done).*" Press the **ENTER** key or the **middle-mouse-button** to indicate you are done picking entities to be deleted.

4. In the prompt window, the message "*OK to delete 1 curve, 1 constraint and 1 dimension? (Yes)*" will appear. The "*1 constraint*" is the *parallel constraint* created by the ***Dynamic Navigator***.



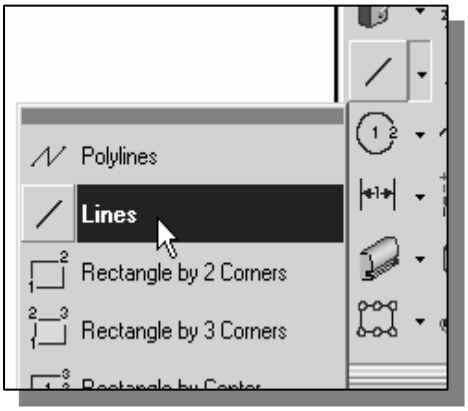
5. Press **ENTER**, or pick **Yes** in the pop-up menu, to delete the selected line. The constraints and dimensions are used as geometric control variables. When the geometry is deleted, the associated control features are also removed.

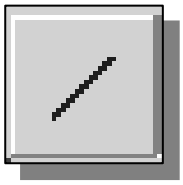
6. In the prompt window, you will see the message "*Pick entity to delete*." By default, *I-DEAS* remains in the **Delete** command and expects you to select additional entities to be erased.

7. Press the **ENTER** key or the **middle-mouse-button** to end the **Delete** command.

Creating a Single Line

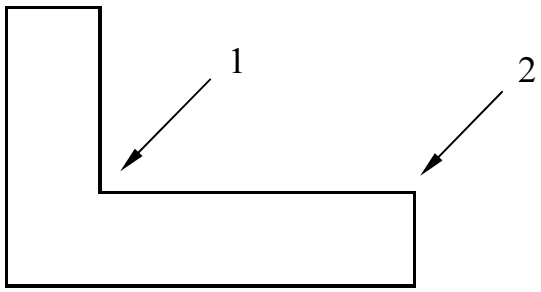
Now we will create a line at the same location by using the **Lines** command.

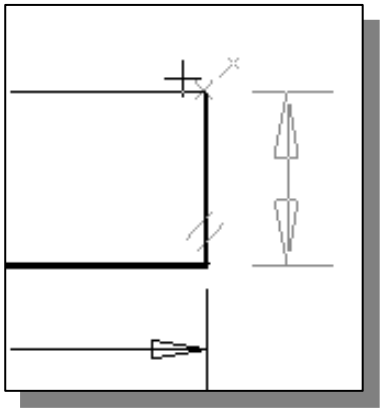




1. Pick **Lines** in the icon panel. (The icon is located in the same stack as the Polylines icon.) Press and hold down the **left-mouse-button** on the Polylines icon to display the available choices. Select the **Lines** command with the **left-mouse-button** when the option is highlighted.

2. The message "*Locate start*" is displayed in the prompt window. Move the graphics cursor near *point 1* and, as the *endpoint* symbol is displayed, pick with the **left-mouse-button**.





3. Move the graphics cursor near *point 2* and click the **left-mouse-button** when the *endpoint* symbol is displayed.

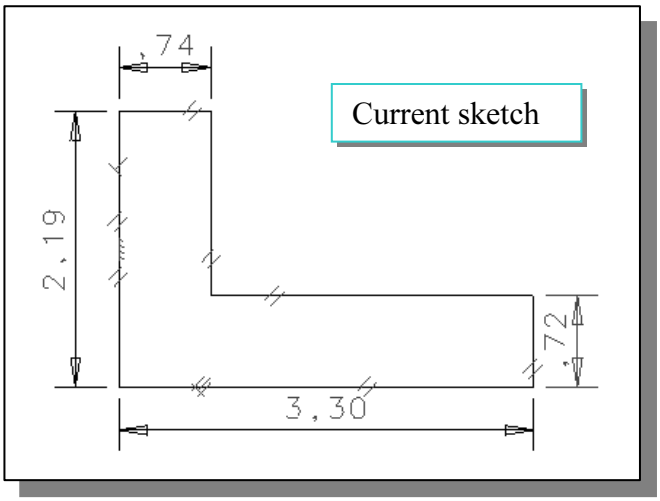
   ➢ Notice the *Dynamic Navigator* creates the parallel constraint and the dimension as the geometry is constructed.

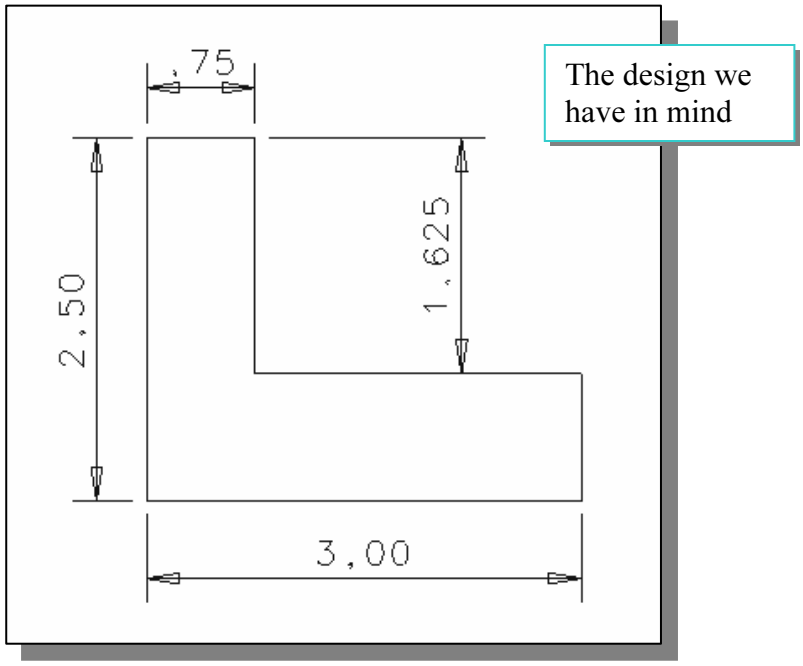
4. The message "*Locate start*" is displayed in the prompt window. Press the **ENTER** key or use the **middle-mouse-button** to end the **Lines** command.

Considerations of Design Intent

While creating the sketch, it is very important to keep in mind the design intent. Always consider functionality of the part and key features of the design. Using *I-DEAS*, we can accomplish and maintain the design intent at all levels of the design process.

The dimensions automatically created by *I-DEAS* might not always match with the designer's intent. For example, in our current design, we may want to use the vertical distance between the first two horizontal lines as a key dimension. Even though it is a very simple calculation to figure out the corresponding length by using the vertical dimension at the far right, for more complex designs it might not be as simple, and to do additional calculations is definitely not desirable. The next section describes re-dimensioning the sketch.





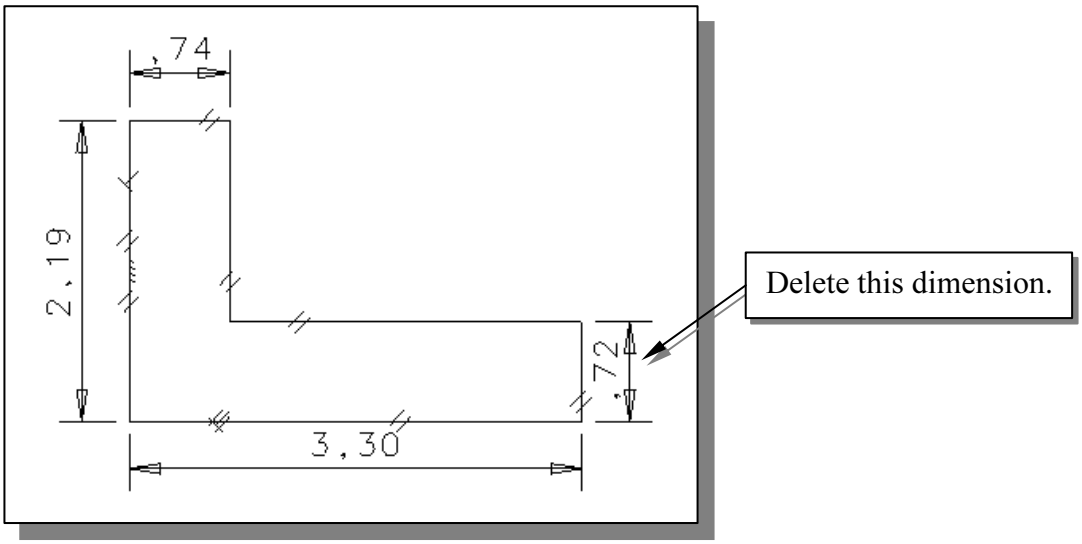
Step 2: Apply/Delete/Modify Constraints and Dimensions

❖ As the sketch is made, *I-DEAS* automatically applies some of the geometric constraints (such as *horizontal*, *parallel* and *perpendicular*) to the sketched geometry. We can continue to modify the geometry, apply additional constraints, and/or define the size of the existing geometry. In this example, we will illustrate deleting existing dimensions and add new dimensions to describe the sketched entities.

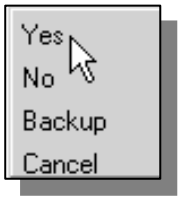
- To maintain our design intent, we will first remove the unwanted dimension and then create the desired dimension.

1. Pick **Delete** in the icon panel. (The icon is located in the last row of the application icon panel.)

2. Pick the dimension as shown.

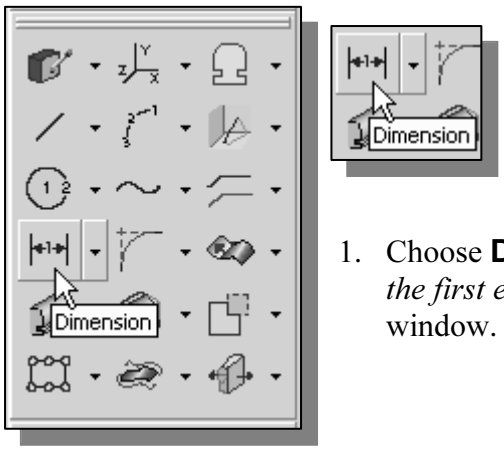


3. Press the **ENTER** key or the **middle-mouse-button** to accept the selection.



4. In the prompt window, the message "*OK to delete 1 dimension?*" is displayed. Pick **Yes** in the popup menu, or press the **ENTER** key or the **middle-mouse-button** to delete the selected dimension. End the **Delete** command by hitting the middle-mouse-button again.

Creating Desired Dimensions

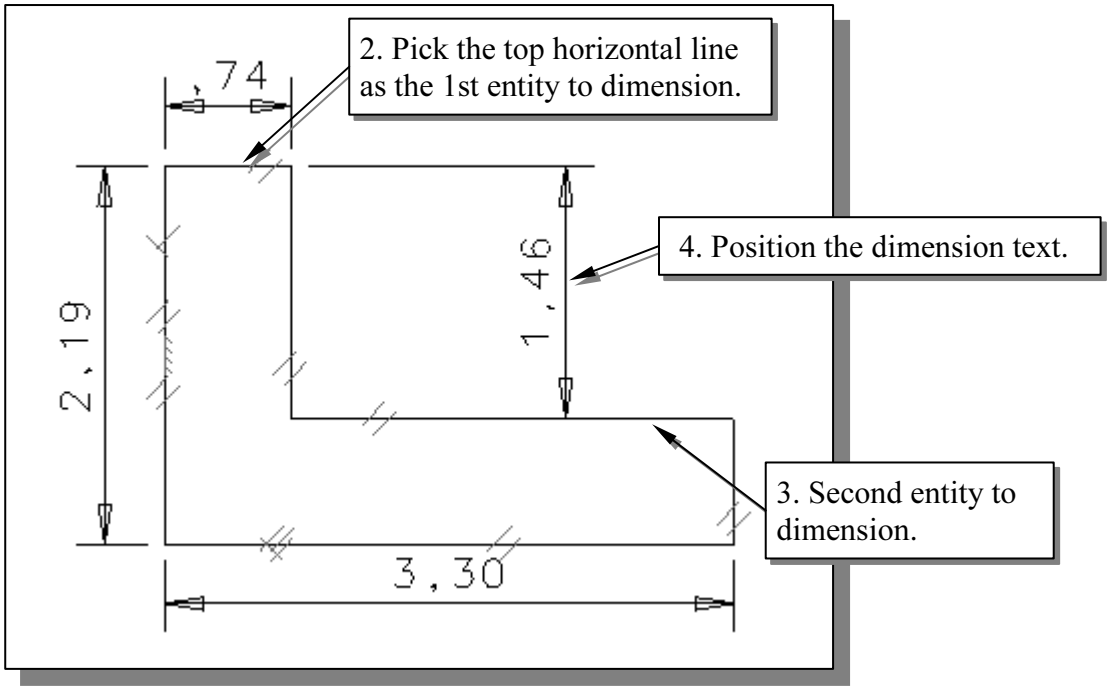


1. Choose **Dimension** in the icon panel. The message "*Pick the first entity to dimension*" is displayed in the prompt window.

2. Pick the **top horizontal line** as shown in the figure below.

3. Pick the **second horizontal line** as shown.

4. Place the text to the right of the model.

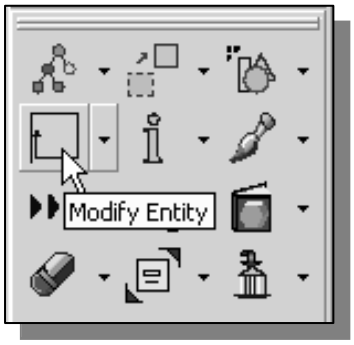


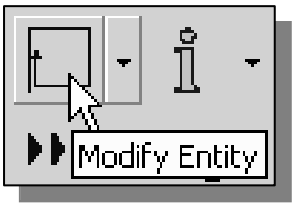
5. Press the **ENTER** key or the **middle-mouse-button** to end the **Dimension** command.

- In *I-DEAS*, the Dimension command will create a linear dimension if two parallel lines are selected (distance in between the two lines). Selecting two lines that are not parallel will create an angular dimension (angle in between the two lines.)

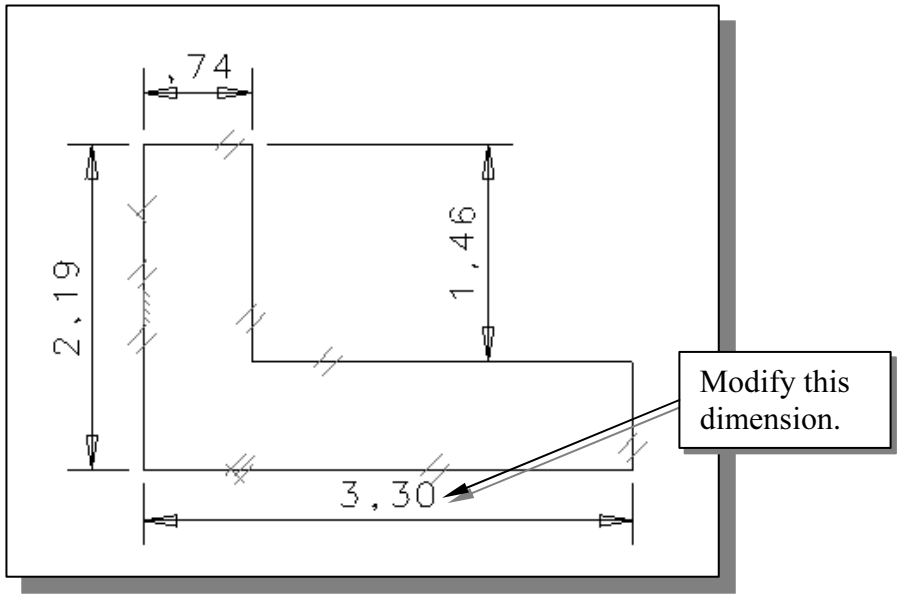
Modifying Dimensional Values

❖ Next we will adjust the dimensional values to the desired values. One of the main advantages of using a feature-based parametric solid modeler, such as *I-DEAS*, is the ability to easily modify existing entities. The operation of modifying dimensional values will demonstrate implementation of the design philosophy of "shape before size." In *I-DEAS*, several options are available to modify dimensional values. In this lesson, we will demonstrate two of the options using the **Modify** command. The Modify command icon is located in the second row of the application icon panel; the icon is a picture of an arrowhead with a long tail.



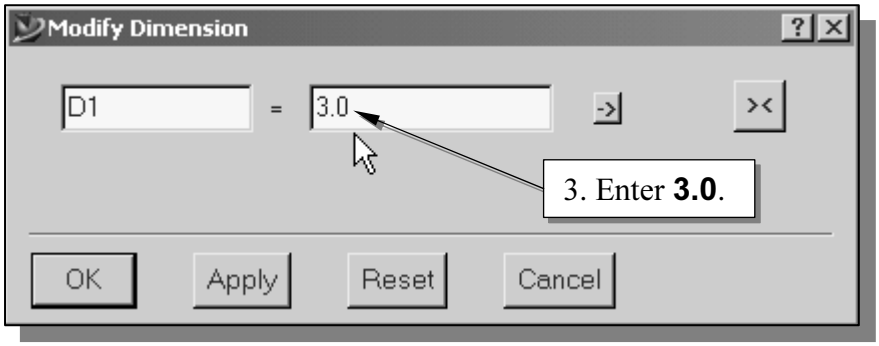


1. Choose **Modify** in the icon panel. (The icon is located in the second row of the application icon panel. If the icon is not on top of the stack, press and hold down the left-mouse-button on the displayed icon, then select the Modify icon.) The message "*Pick entity to modify*" is displayed in the prompt window.



2. Pick the dimension as shown (the number might be different than displayed). The selected dimension will be highlighted. The *Modify Dimension* window appears.

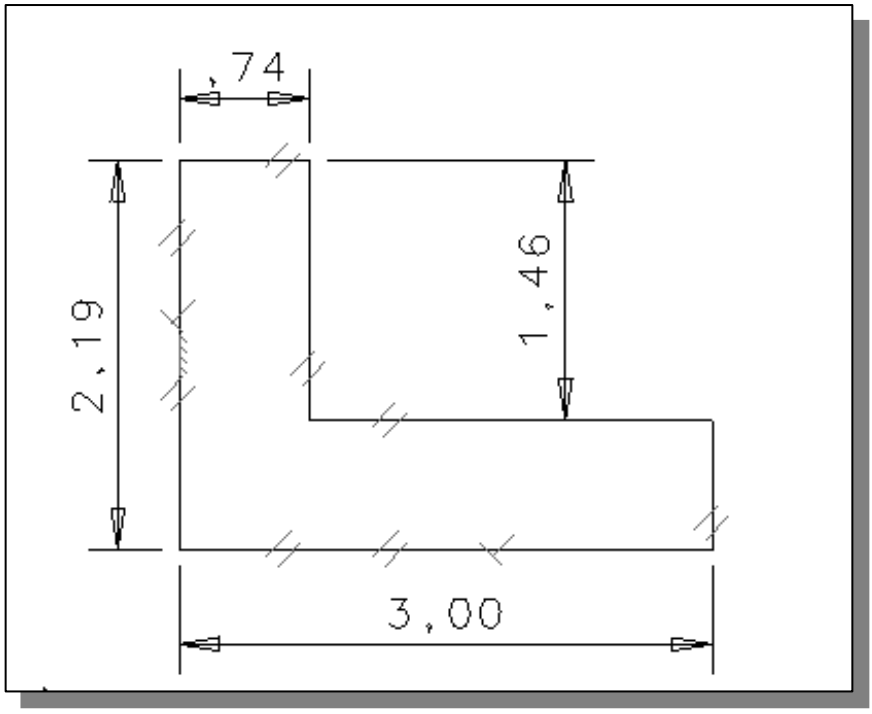
❖ In the *Modify Dimension* window, the value of the selected dimension is displayed and also identified by a *name* in the format of "Dxx," where the "D" indicates it is a dimension and the "xx" is a number incremented automatically as dimensions are added. You can change both the name and the value of the dimension by clicking and typing in the appropriate boxes.



3. Type in **3.0** to modify the dimensional value as shown in the above figure.

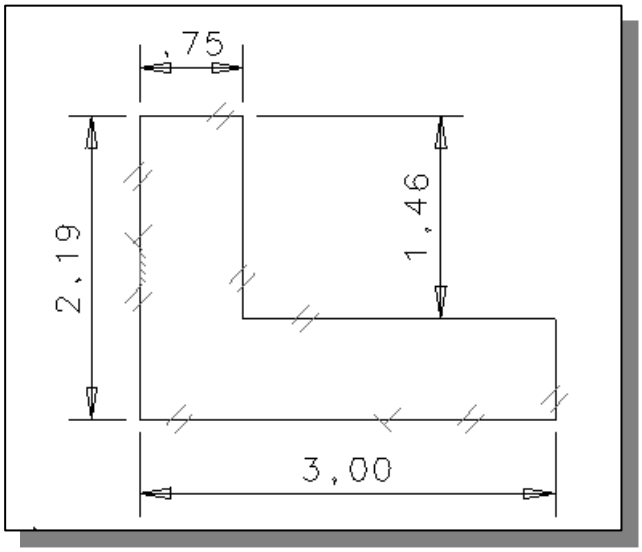
4. Click on the **OK** button to accept the value you have entered.

➢ *I-DEAS* will adjust the size of the object based on the new value entered.



5. On your own, click on the top horizontal dimension and adjust the dimensional value to **0.75**.

6. Press the **ENTER** key or the **middle-mouse-button** to end the **Modify Dimension** command.



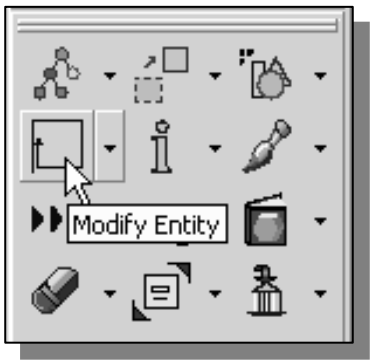
❖ The size of our design is automatically adjusted by *I-DEAS* based on the dimensions we have entered. *I-DEAS* uses the dimensional values as control variables and the geometric entities are modified accordingly. This approach of rough sketching the shape of the design first then finalizing the size of the design is known as the "**shape before size**" approach.

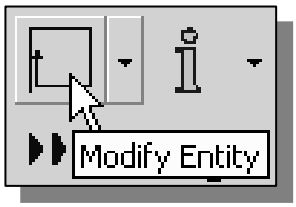
Pre-selection of Entities

I-DEAS provides a flexible graphical user interface that allows users to select graphical entities BEFORE the command is selected (*pre-selection*), or AFTER the command is selected (*post-selection*). The procedure we have used so far is the *post-selection* option. To pre-select one or more items to process, hold down the **SHIFT** key while you pick. Selected items will stay highlighted. You can *deselect* an item by selecting the item again. The item will be toggled on and off by each click. Another convenient feature of pre-selection is that the selected items remain selected after the command is executed.

1. Pre-select all of the dimensions by holding down the **SHIFT** key and clicking the **left-mouse-button** on each dimension value.

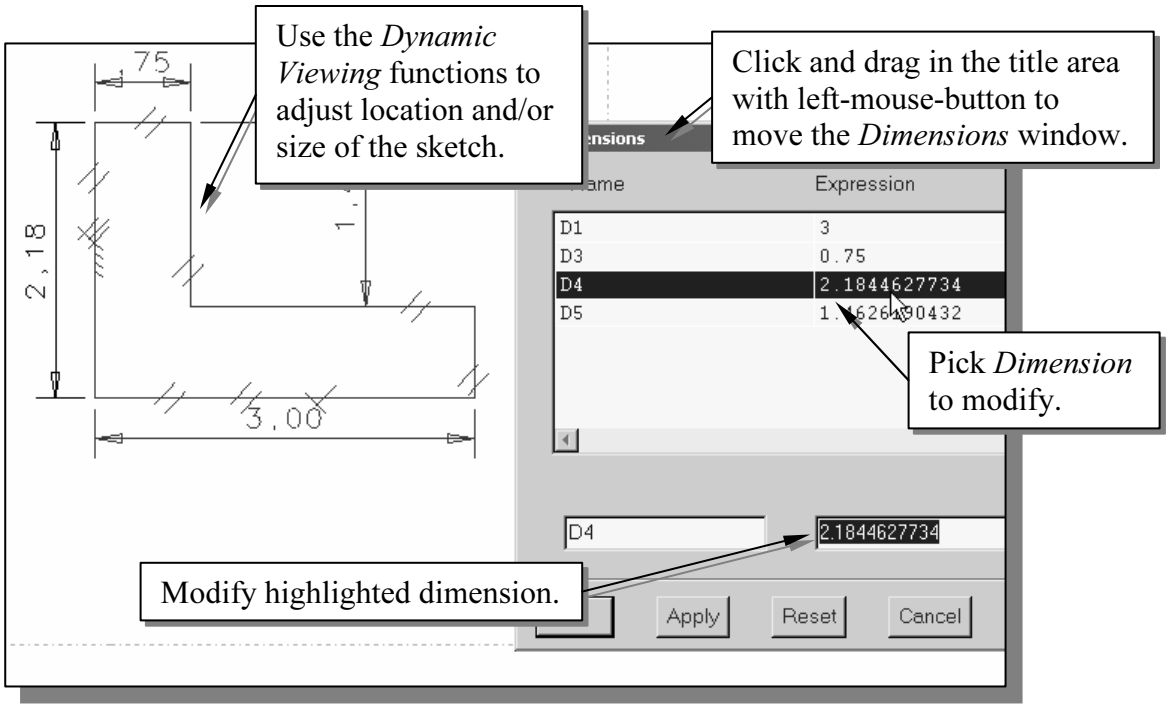
PRE-SELECT SHIFT + LEFT-mouse-button

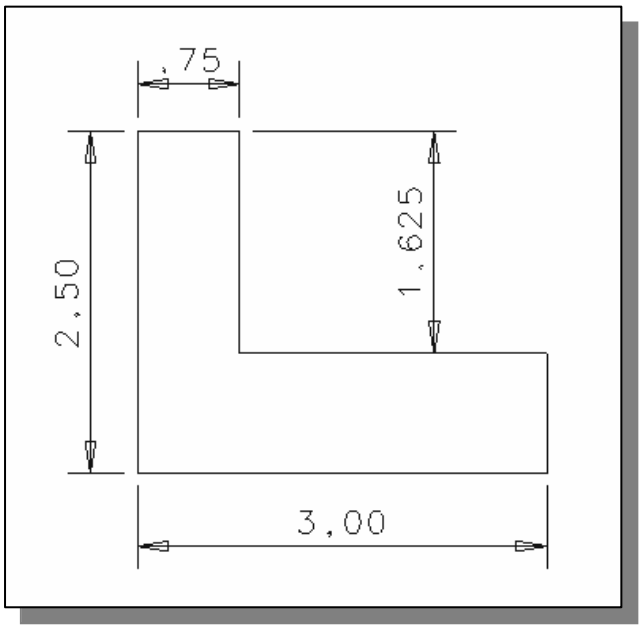




2. Select the **Modify** icon. The *Dimensions* window appears.

3. Move the *Dimensions* window around so that it does not overlap the part drawing. Do this by "clicking and dragging" the window's title area with the left-mouse-button. You can also use the *Dynamic Viewing* functions (activate the graphics window first) to adjust the scale and location of the entities displayed in the graphics window (F1 and the mouse; F2 and the mouse).





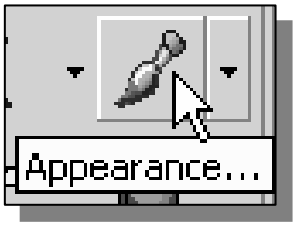
4. Click on one of the dimensions in the pop-up window. The selected dimension will be highlighted in the graphics window. Type in the desired value for the selected dimension. **DO NOT** hit the **ENTER** key. Select another dimension from the list to continue modifying. Modify all of the dimensional values to the values as shown.

5. Click the **OK** button to accept the values you have entered and close the *Dimensions* window.

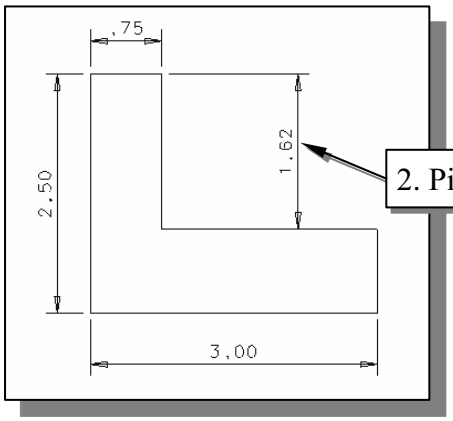
- *I-DEAS* will now adjust the size of the shape to the desired dimensions. The design philosophy of "shape before size" is implemented quite easily. The geometric details are taken care of by *I-DEAS*.

Changing the Appearance of Dimensions

- The right vertical dimension we modified is displayed as 1.62, instead of the entered value (1.625.) We can adjust the appearance of dimensions by using the **Appearance** command.

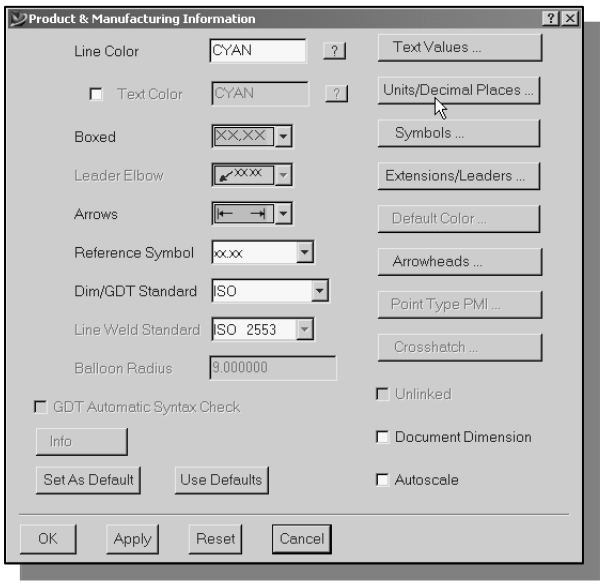


1. Choose **Appearance** in the icon panel. (The icon is located in the second row of the application icon panel. If the icon is not on top of the stack, press and hold down the left-mouse-button on the displayed icon, then select the Appearance icon.)



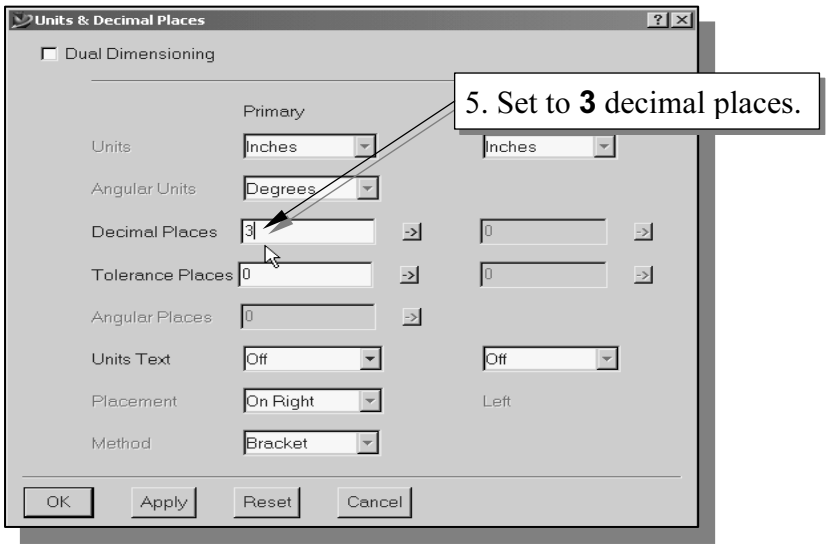
2. The message "*Pick entity to modify*" is displayed in the prompt window. Pick the right vertical dimension as shown in the figure.

3. The message "*Pick entity to modify (Done)*" is displayed in the prompt window. Press the **ENTER** key or use the **middle-mouse-button** to accept the selected object.



4. In the *Product & Manufacturing Information* window, click on the **Units/Decimal Places…**button. The *Units & Decimal Places* window appears.

5. Set the decimal places to **3** to display three digits after the decimal point.

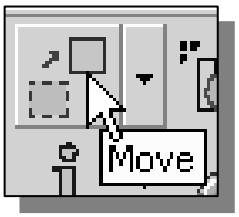


6. Click on the **OK** button to exit the *Units & Decimal Places* window.

7. Click on the **OK** button to exit the *Product & Manufacturing Information* window.

8. Press the **ENTER** key or the **middle-mouse-button** to end the Appearance command.

Repositioning Dimensions



1. Choose **Move** in the icon panel. (The icon is located in the first row of the application icon panel.) The message "*Pick entity to move*" is displayed in the prompt window.

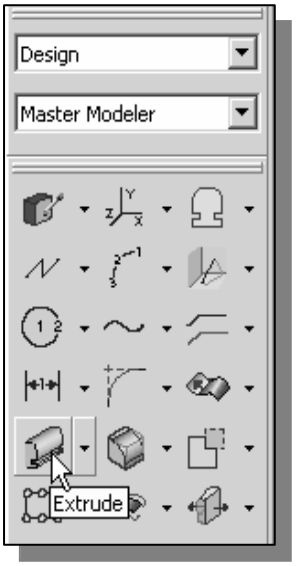
2. Select any of the dimensions displayed on the screen.

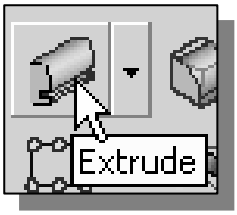
3. Move the cursor to position the dimension in a new location. Left-click once to accept the new location.

4. Press the **ENTER** key or the **middle-mouse-button** to end the Move command.

Step 3: Completing the Base Solid Feature

- Now that the 2D sketch is completed, we will proceed to the next step: create a 3D feature 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.

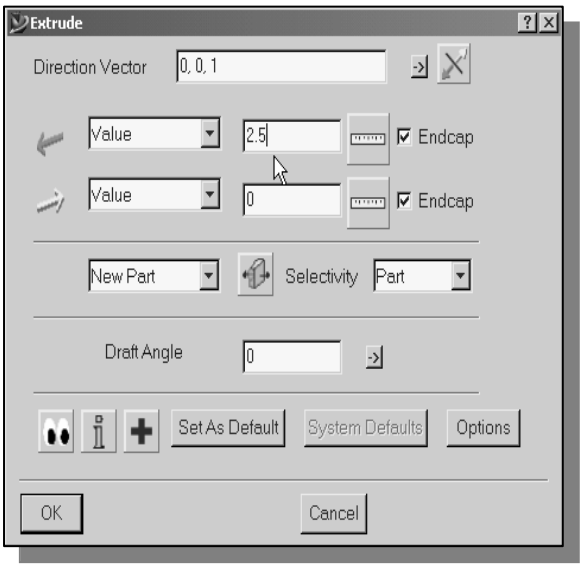




1. Choose **Extrude** in the icon panel. The **Extrude** icon is located in the fifth row of the task specific icon panel. Press and hold down the left-mouse-button on the icon to display all the choices. If a different choice were to be made, you would slide the mouse up and down to switch between different options. In the prompt window, the message "*Pick curve or section*" is displayed.

2. Pick any edge of the 2D shape. By default, the Extrude command will automatically select all segments of the shape that form a closed region. Notice the different color signifying the selected segments.

3. Notice the *I-DEAS* prompt "*Pick curve to add or remove. (Done)*" We can select more geometric entities or deselect any entity that has been selected. Picking the same geometric entity will again toggle the selection of the entity "on" or "off" with each left-mouse-button click. Press the **ENTER** key to accept the selected entities.



4. The *Extrude Section* window will appear on the screen. Enter **2.5**, in the first value box, as the *extrusion distance* and confirm that the ***New part*** option is set as shown in the figure.

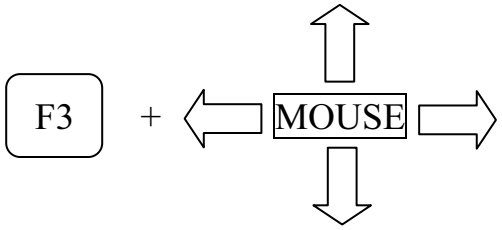
5. Click on the **OK** button to accept the settings and extrude the 2D section into a 3D solid.

➢ Notice all of the dimensions disappeared from the screen. All of the dimensional values and geometric constraints are stored in the database by *I-DEAS* and they can be brought up at any time.

Display Viewing Commands

❖ 3D Dynamic Rotation – (1) F3 and the mouse

The *I-DEAS Dynamic Viewing* feature allows users to do "*real-time*" rotation of the display. Hold the **F3** function key down and move the mouse to rotate the display. This allows you to rotate the displayed model about the screen's X (horizontal), Y (vertical), and Z (perpendicular to the screen) axes. Start with the cursor near the center of the screen and hold down **F3**; moving the cursor up or down will rotate about the screen X-axis while moving the cursor left or right will control the rotation about the screen Y-axis. Start with the cursor in the corner of the screen and hold down **F3**, which will control the rotation about the screen Z-axis.

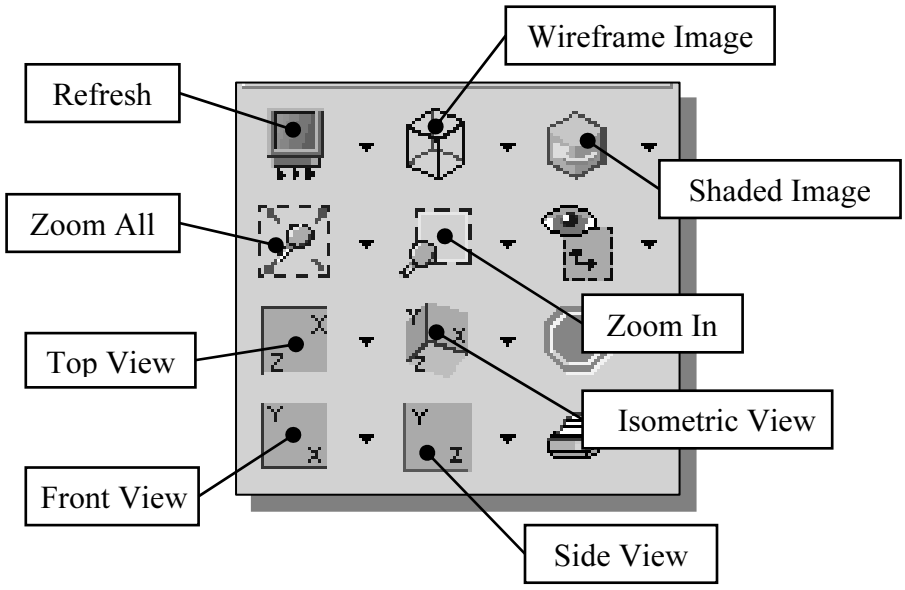


❖ 3D Dynamic Rotation – (2) The middle mouse button

Holding the middle mouse button and dragging with the mouse also allow us to rotate the display.

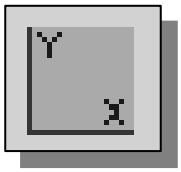
Display Icon Panel

The *display* icon panel contains various icons to handle different viewing operations. These icons control the screen display, such as the view scale, the view angle, redisplay, and shaded and hidden line displays.

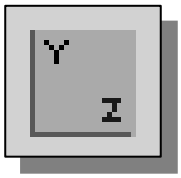


View Icons:

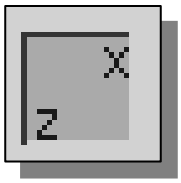
Front, Side, Top, Bottom, Isometric, and Perspective: These six icons are the standard view icons. Selecting any of these icons will change the viewing angle. Try each one as you read its description below:



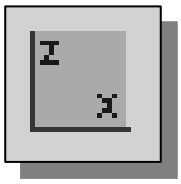
Front View (X-Y Workplane)



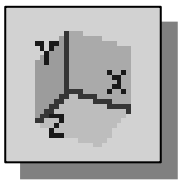
Right Side View



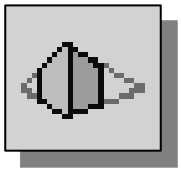
Top View



Bottom View



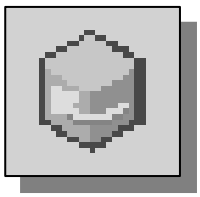
Isometric View



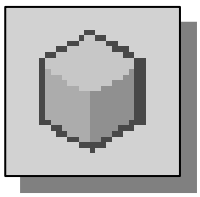
Perspective View

❖ Shaded Solids:

Depending on your display type, you will pick either Shaded Hardware or Shaded Software to get shaded images of 3D objects. **Shaded Hardware** on a workstation with OGL display capability allows real-time dynamic rotation (**F3** and the mouse) of the shaded 3D solids. A workstation with X3D display capability allows the use of the **Shaded Software** command to get the shaded image without the real-time dynamic rotation capability.

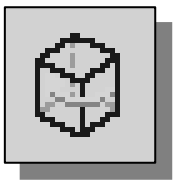


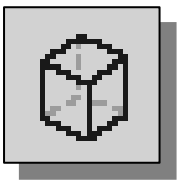
Shaded Hardware

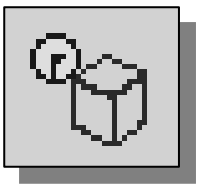


Shaded Software

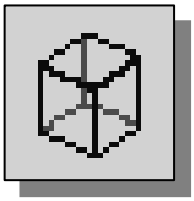
❖ **Hidden-line Removal:**
Three options are available to generate images with all the back lines removed.


Hidden Hardware

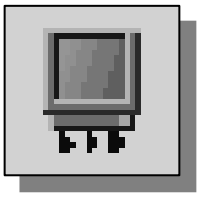

Precise Hidden

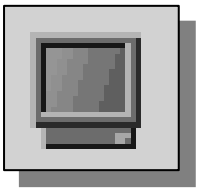

Quick Hidden

❖ **Wireframe Image:**
This icon allows the display of the 3D objects using the basic wireframe representation scheme.

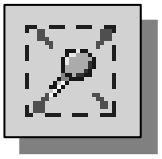

Wireframe

❖ **Refresh and Redisplay:**
Use these commands to regenerate the graphics window.

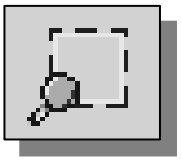

Refresh


Redisplay

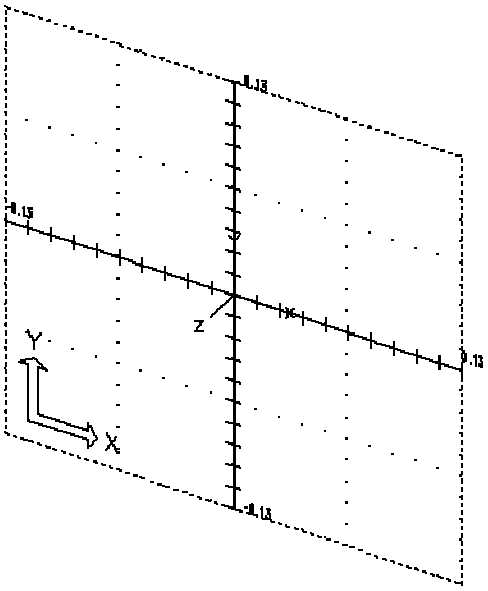
❖ **Zoom-All:**
Adjust the viewing scale factor so that all objects are displayed.


Zoom-All

❖ **Zoom-In:**
Allows the users to define a rectangular area, by selecting two diagonal corners, which will fill the graphics window.


Zoom-In

Workplane – 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. In using a geometric modeler, therefore, we 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 results to expect based upon 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; 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** or **User Coordinate Systems** relative to the world coordinate system. Once a local 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 input and output is done in a 2D Cartesian system. Typical input devices such as a mouse or digitizer 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 the 2D picture plane (monitor).

The *I-DEAS* ***workplane*** is a special construction tool that enables the planar nature of 2D input devices to be directly mapped into the 3D coordinate system. The workplane 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 workplane is aligned to the world coordinate system.

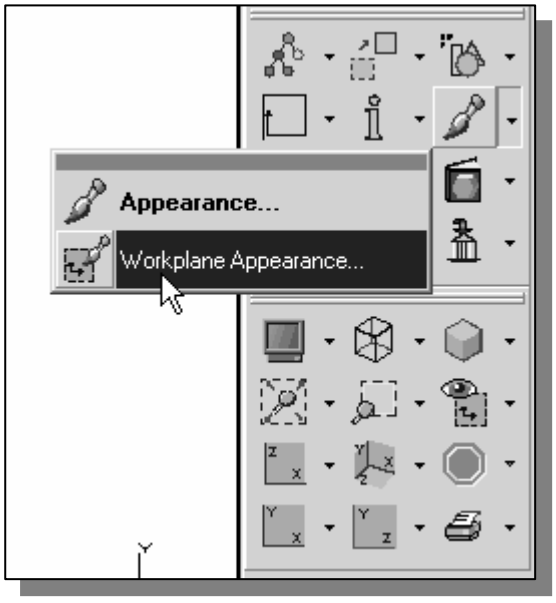
The basic design process of creating solid features in the *I-DEAS* task is a three-step process:

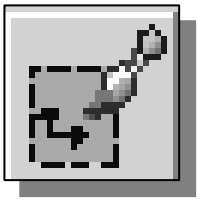
1. Select and/or define the workplane.
2. Sketch and constrain 2D planar geometry.
3. Create the solid feature.

These steps can be repeated as many times as needed to add additional features to the design. The base feature of the *L-Block* model was created following this basic design process; we used the default settings where the workplane is aligned to the world coordinate system. We will next add additional features to our design and demonstrate how to manipulate the *I-DEAS* workplane.

Workplane Appearance

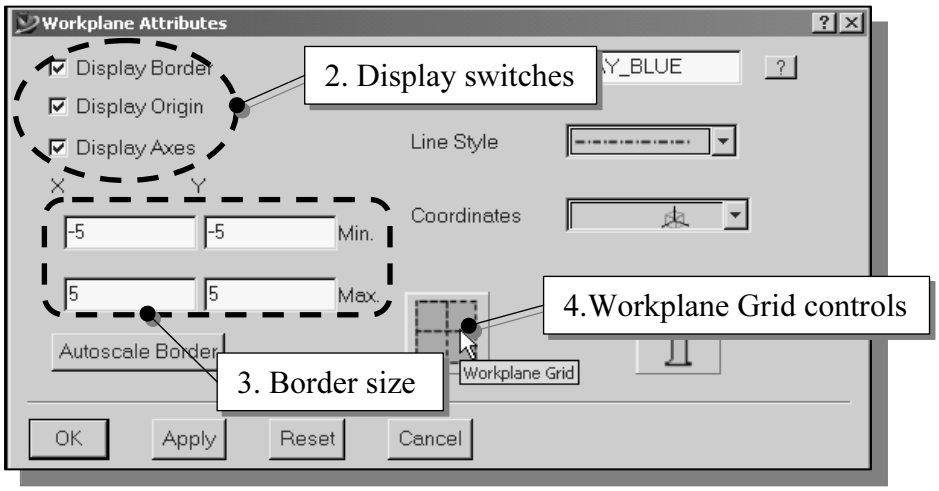
The workplane is a construction tool; it is a coordinate system that can be moved in space. The size of the workplane display is only for our visual reference, since we can sketch on the entire plane, which extends to infinity.





1. Choose **Workplane Appearance** in the icon panel. (The icon is located in the second row of the application icon panel. If the icon is not on top of the stack, press and hold down the left-mouse-button on the displayed icon to display all the choices, then select the Workplane Appearance icon.) The *Workplane Attributes* window appears.

2. Toggle ***on*** the three display switches as shown.

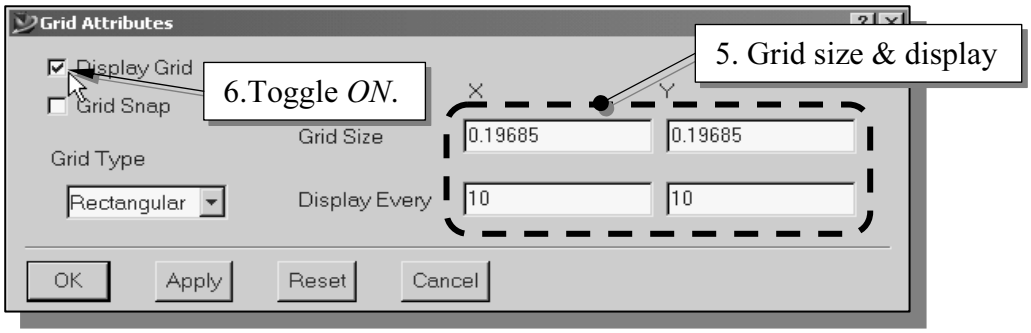


3. Adjust the **workplane border size** by entering the *Min*. and *Max*. values as shown.

4. In the *Workplane Attributes* window, click on the **Workplane Grid** button. The *Grid Attributes* window appears.

5. Change the ***Grid Size*** settings by entering the values as shown.

6. Toggle ***on*** the ***Display Grid*** option if it is not already switched on.



❖ Although the *Grid Snap* option is available, its usage in parametric modeling is not recommended. The *Grid Snap* concept does not conform to the "*shape before size*" philosophy and most real designs rarely have uniformly spaced dimension values.

7. Pick **Apply** to view the effects of the changes.

8. Click on the **OK** button to exit the *Grid Attributes* window.

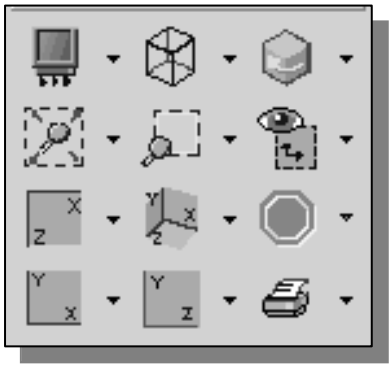
9. Click on the **OK** button to exit the *Workplane Attributes* window.

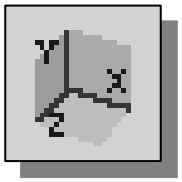
10. On your own, use [**F3**+**Mouse**] to dynamically rotate the part and observe the workplane is aligned with the surface corresponding to the first sketch drawn.

Step 4: Adding additional features

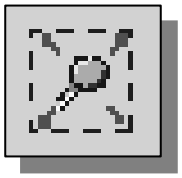
➢ Sketch In Place

One option to manipulate the workplane is with the **Sketch in Place** command. The Sketch in Place command allows the user to sketch on an existing part face. The workplane is reoriented and is attached to the face of the part.

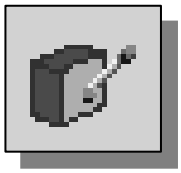




1. Choose **Isometric View** in the display viewing icon panel.

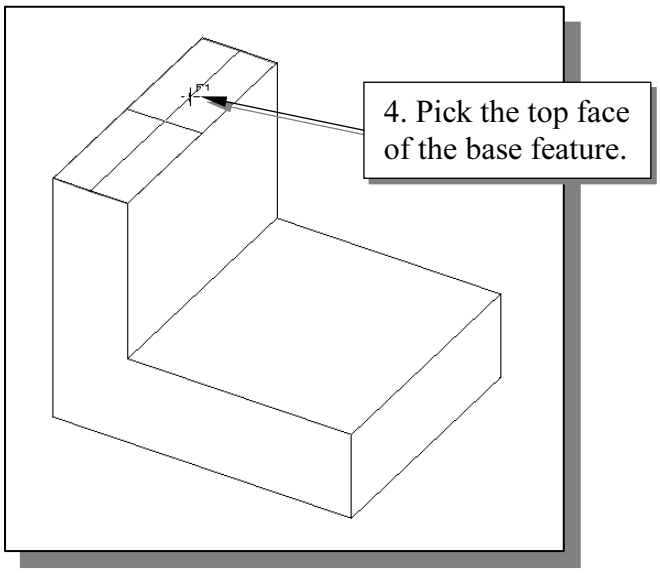


2. Choose **Zoom-All** in the display viewing icon panel.



3. Choose **Sketch in Place** in the icon panel. In the prompt window, the message "*Pick plane to sketch on*" is displayed.

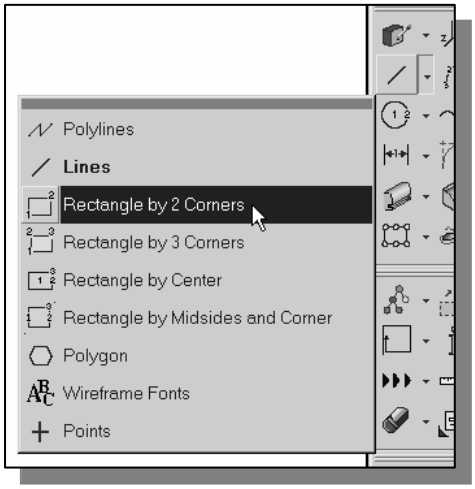
4. Pick the top face of the horizontal portion of the 3D object by left-clicking the surface, when it is highlighted as shown in the figure below.

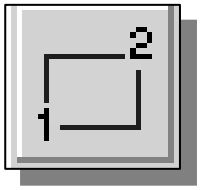


❖ Notice that, as soon as the top surface is picked, *I-DEAS* automatically orients the *workplane* to the selected surface. The surface selected is highlighted with a highlighted color to indicate the attachment of the *workplane*.

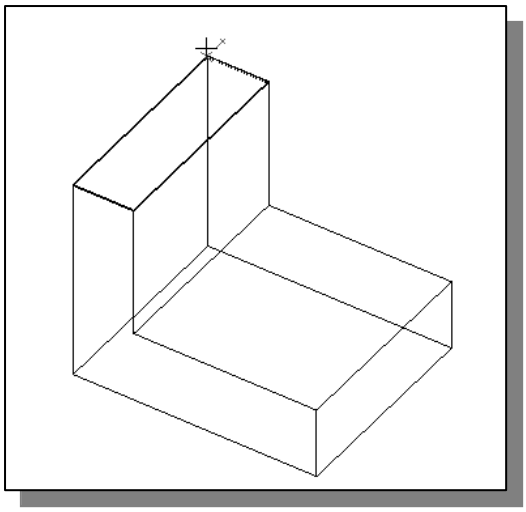
Step 4-1: Adding an extruded feature

➢ Next, we will create another 2D sketch, which will be used to create an extruded feature that will be added to the existing solid object.

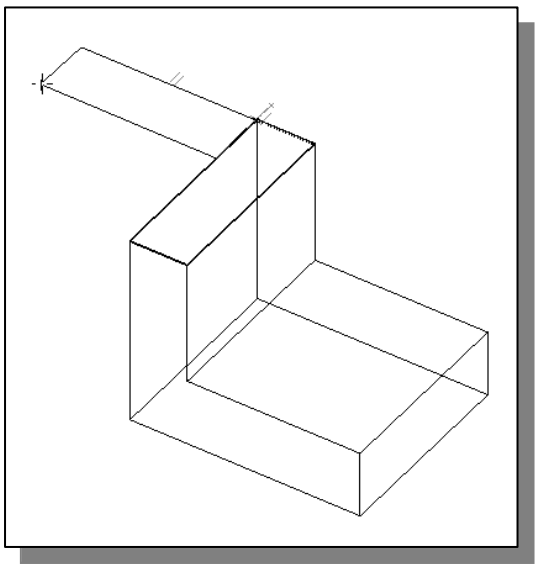




1. Choose **Rectangle by 2 Corners** in the icon panel. This command requires the selection of two locations to identify the two opposite corners of a rectangle. The message "*Locate first corner*" is displayed in the prompt window.

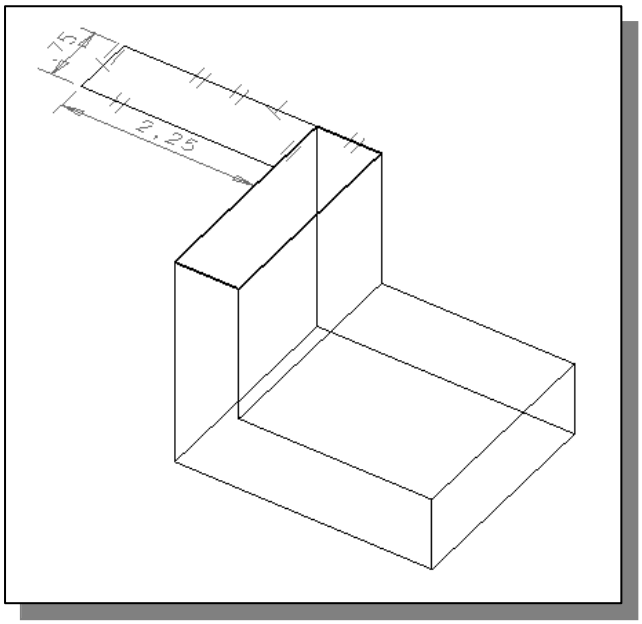


2. Create a rectangle by first selecting the top-left corner of the solid model as shown in the figure. Note that *I-DEAS* automatically snaps to the end points of existing geometry.

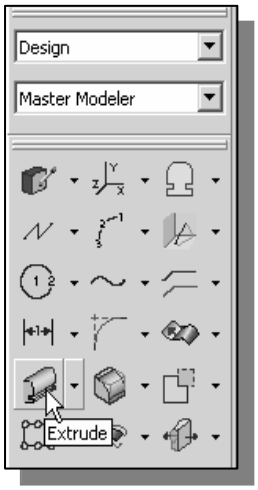


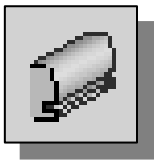
3. Create a rectangle of arbitrary size by selecting a location that is toward the front left direction of the last location as shown in the figure.

➢ Note that *I-DEAS* automatically applies dimensions as the rectangle is constructed. Do not be concerned with the actual numbers of the dimensions, which we will adjust in the next section.



4. On your own, modify the two dimensions to **0.75** and **2.25** as shown in the figure.

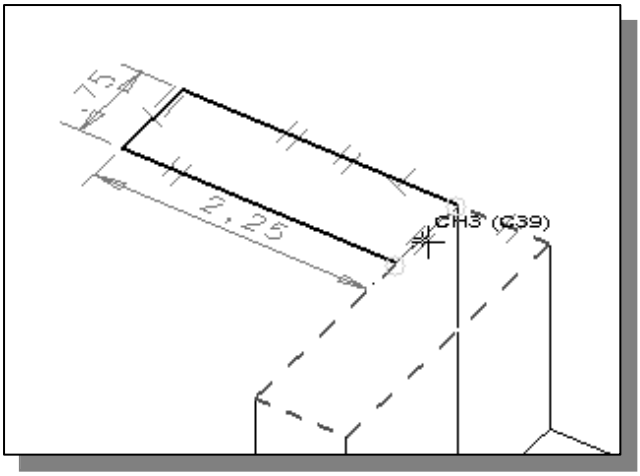




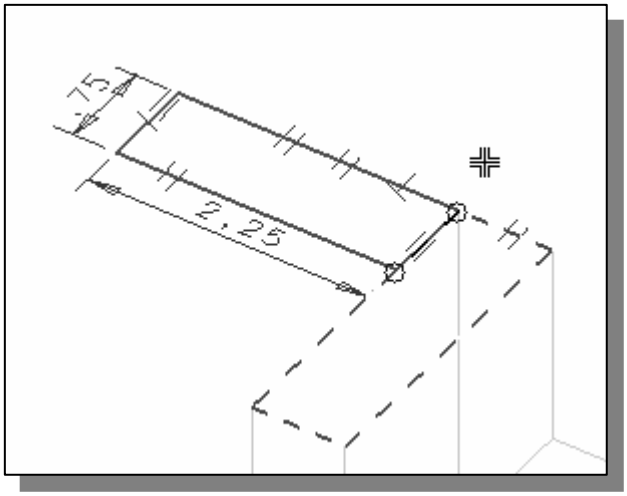
5. Choose **Extrude** in the icon panel. The Extrude icon is located in the fifth row of the task specific icon panel.

6. In the prompt window, the message "*Pick curve or section*" is displayed. Pick the front edge of the 2D rectangle we just created. By default, the Extrude command will automatically select all neighboring segments of the selected segment to form a closed region. Notice the different color signifying the selected segments.

7. Pick the segment in between the displayed two small circles so that the highlighted entities form a closed region.

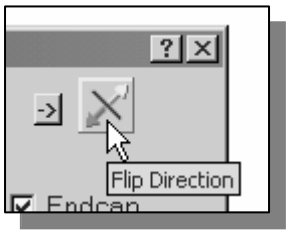


8. The short segment of the sketched rectangle, aligned to the top edge of the solid model, is highlighted and notice the double line cursor is displayed. Press the **ENTER** key once, or click once with the **middle-mouse-button**, to accept the selected entity.

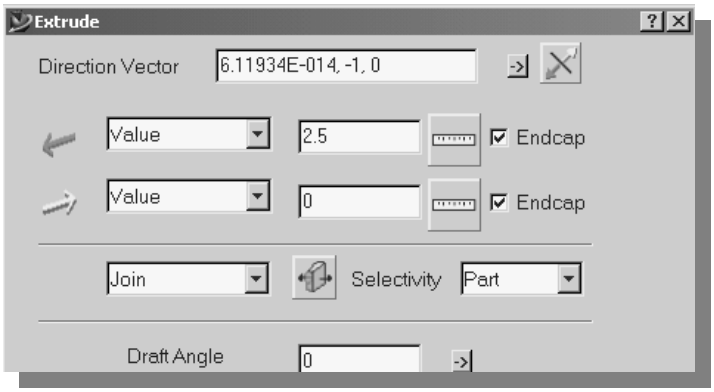


❖ Attempting to select a line where two entities lie on top of one another (i.e. coincide) causes confusion as indicated by the double line cursor ╬ symbol and the prompt window message "*Pick curve to add or remove (Accept)***". This message indicates *I-DEAS* needs you to confirm the selected item. If the correct entity is selected, you can continue to select additional entities. To reject an erroneously selected entity, press the [**F8**] key to select a neighboring entity or press the right-mouse-button and highlight **Deselect All** from the popup menu.

9. Confirm the four sides of the sketched rectangle are highlighted and press the **ENTER** key once, or click once with the **middle-mouse-button**, to proceed with the Extrude command.



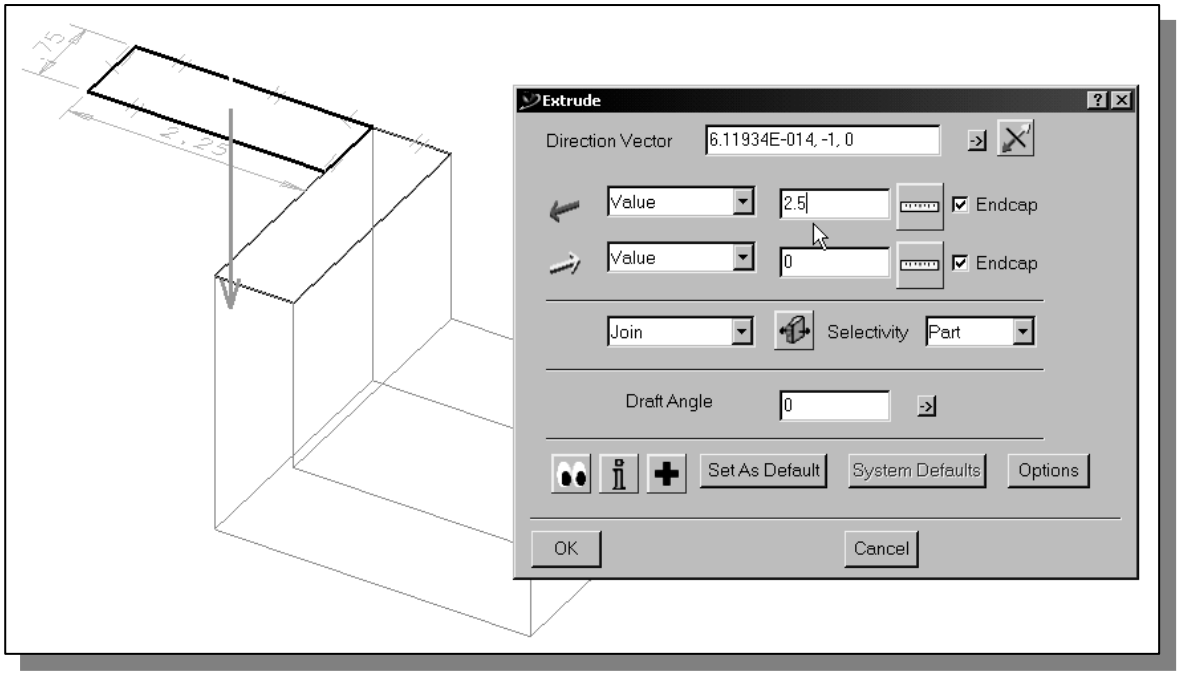
10. The *Extrude* window appears on the screen. Click on the **Flip Direction** button near the upper right corner of the *Extrude* window to switch the extrusion direction so that the green arrow points downward.



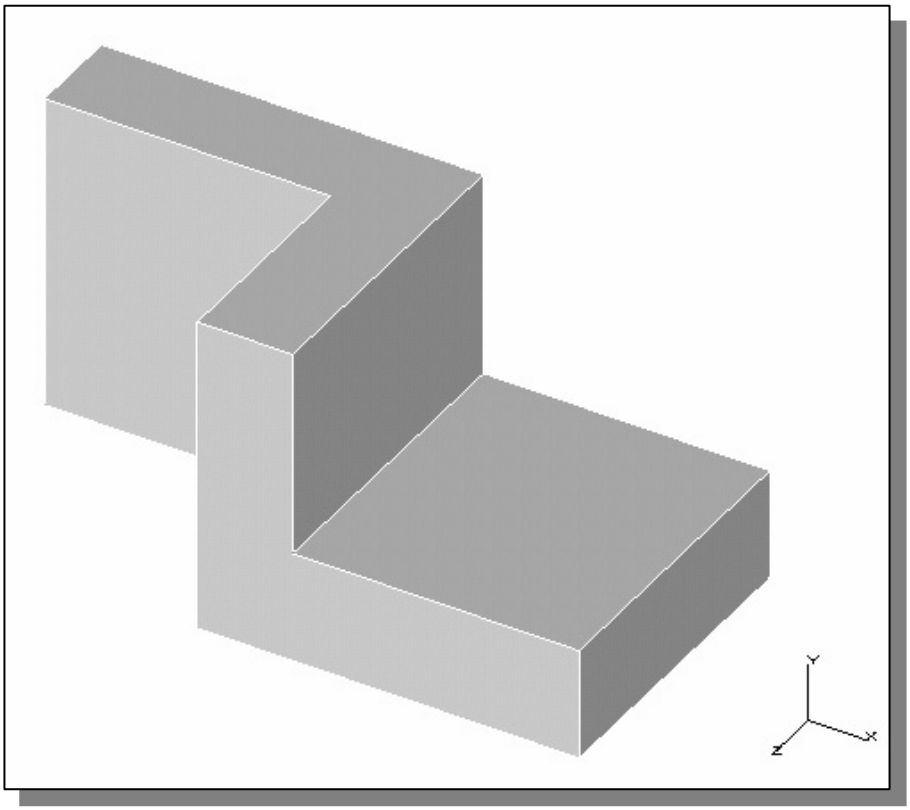
11. Enter **2.5**, in the first value box, as the *extrusion distance*.

12. Confirm that the ***Join*** option is set as shown in the figure.

13. Confirm the extrusion options inside the *Extrude* window and the displayed image inside the *graphics* window are set as shown.

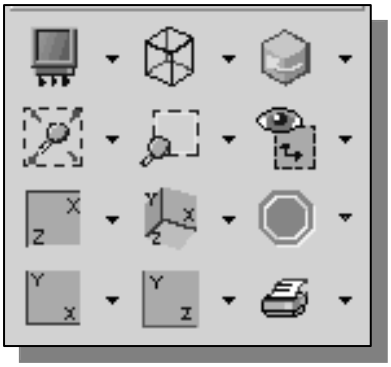


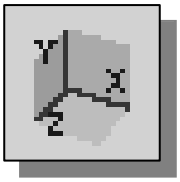
14. Click on the **OK** button to accept the settings and extrude the sketched 2D section into a 3D solid feature of the solid model.



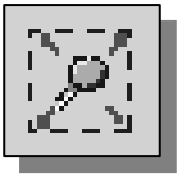
Step 4-2: Adding a Cut Feature

- Next, we will create a **circular cut** feature to the existing solid object.

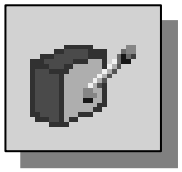




1. Choose **Isometric View** in the display viewing icon panel.

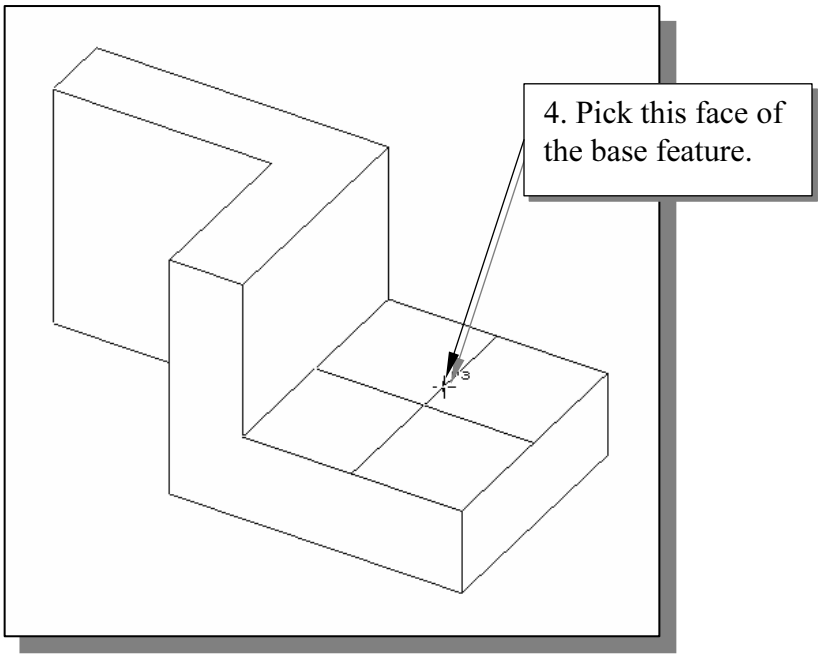


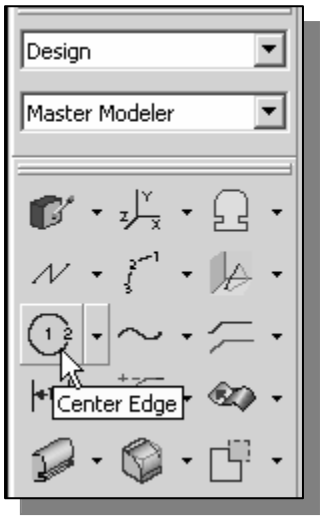
2. Choose **Zoom-All** in the display viewing icon panel.

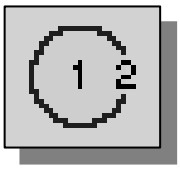


3. Choose **Sketch in Place** in the icon panel. In the prompt window, the message "*Pick plane to sketch on*" is displayed.

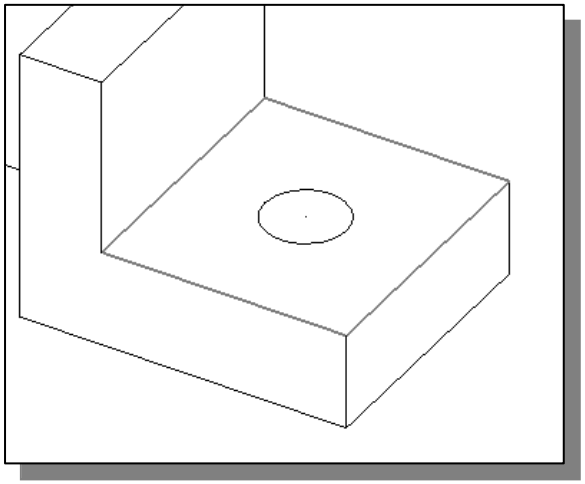
4. Pick the top face of the horizontal portion of the 3D object by left-clicking the surface, when it is highlighted as shown in the below figure.





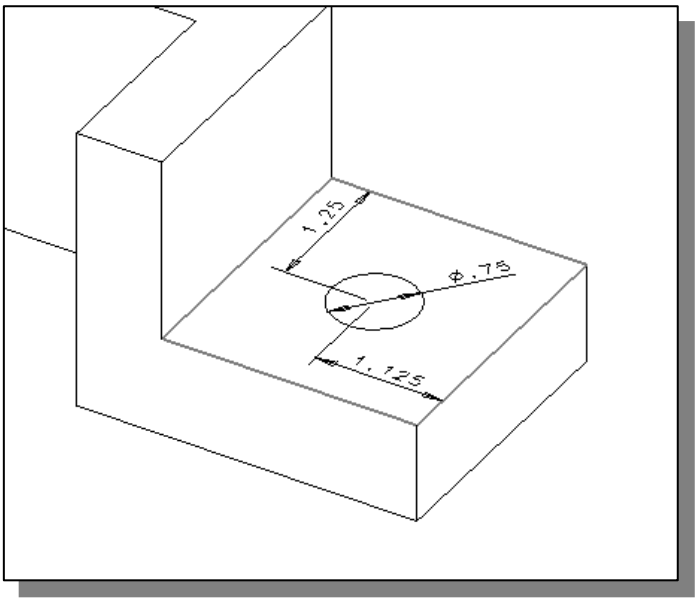


5. Choose **Circle – Center Edge** in the icon panel. This command requires the selection of two locations: first the location of the center of the circle and then a location where the circle will pass through.

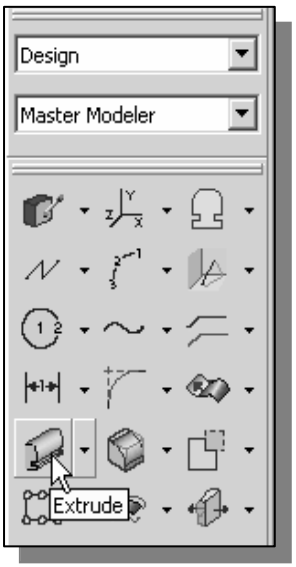


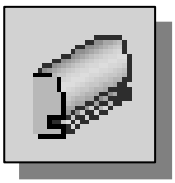
6. On your own, create a circle inside the horizontal face of the solid model as shown.

7. On your own, create and modify the three dimensions as shown.



♦ Extrusion – Cut Option

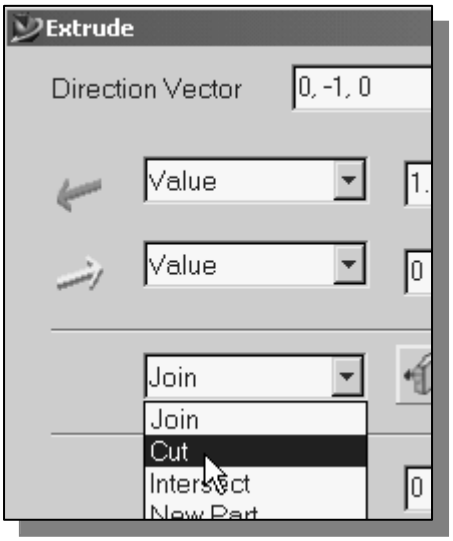




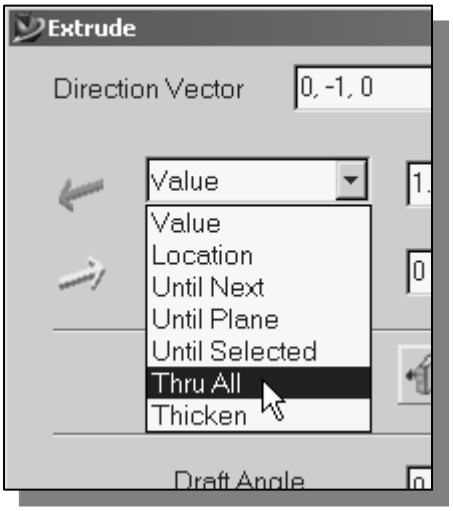
1. Choose **Extrude** in the icon panel. The Extrude icon is located in the fifth row of the task specific icon panel.

2. In the prompt window, the message "*Pick curve or section*" is displayed. Pick the newly sketched circle.

3. At the *I-DEAS* prompt "*Pick curve to add or remove (Done)*," press the **ENTER** key or the **middle-mouse-button** to accept the selection.



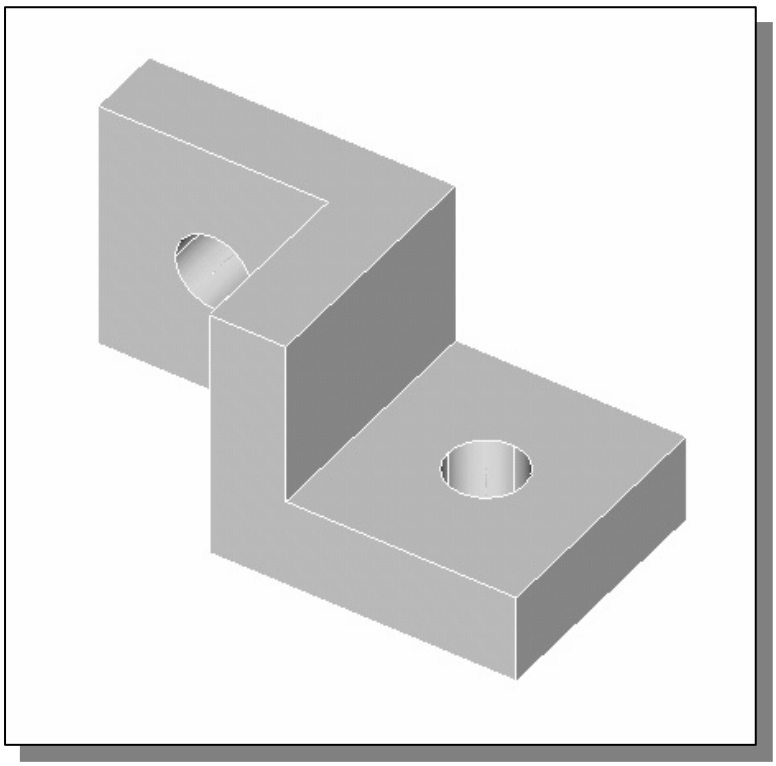
4. The *Extrude Section* window appears. Set the *extrude option* to ***Cut***. Note the extrusion direction displayed in the graphics window.



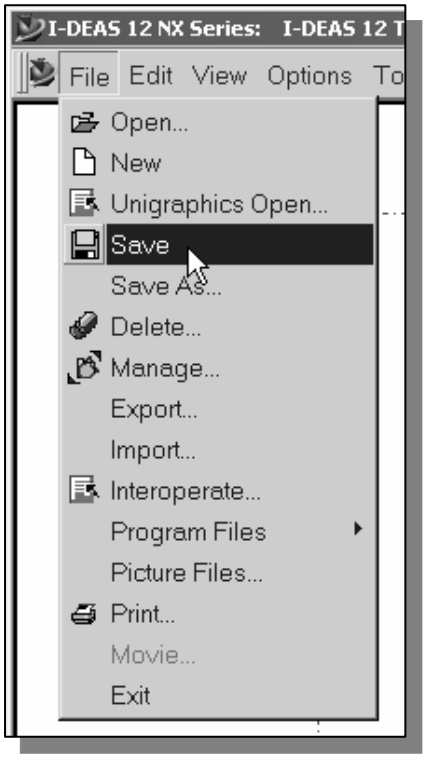
5. Click and hold down the left-mouse-button on the **depth** menu and select the ***Thru All*** option. *I-DEAS* will calculate the distance necessary to cut through the part.

6. Click on the **OK** button to accept the settings. The rectangle is extruded and the front corner of the 3D object is removed.

7. On your own, create the other circular cut feature (same dimensions as the previous feature) on the vertical section and complete the model as shown.



Save the Part and Exit *I-DEAS*



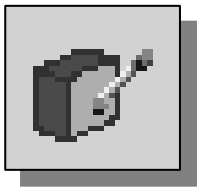
1. From the icon panel, select the **File** pull-down menu. Pick the **Save** option. Notice that you can also use the **Ctrl-S** combination (pressing down the Ctrl key and hitting the "S" key once) to save the part. A small watch appears to indicate passage of time as the part is saved.

2. Now you can leave *I-DEAS*. Use the left-mouse-button to click on **File** in the toolbar menu and select **Exit** from the pull-down menu. A pop-up window will appear with the message "*Save changes before exiting?*" Click on the **NO** button since we have saved the model already.

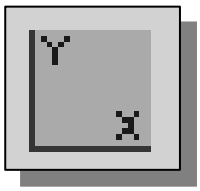
Questions:

1. Describe the “shape before size” design philosophy.
2. How does the *I-DEAS Dynamic Navigator* assist us in sketching?
3. Which command can we use to reposition and align dimensions?
4. How do we modify more than one dimension at a time?
5. What is the difference between the Lines and Polylines commands?
6. How do we change the number of decimal places displayed in dimensions?
7. Identify and describe the following commands:

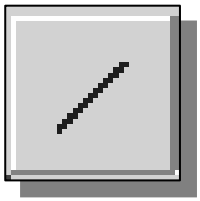
(a)



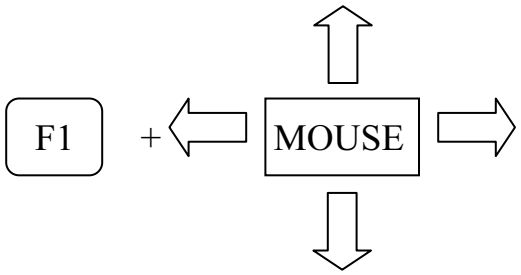
(b)



(c)

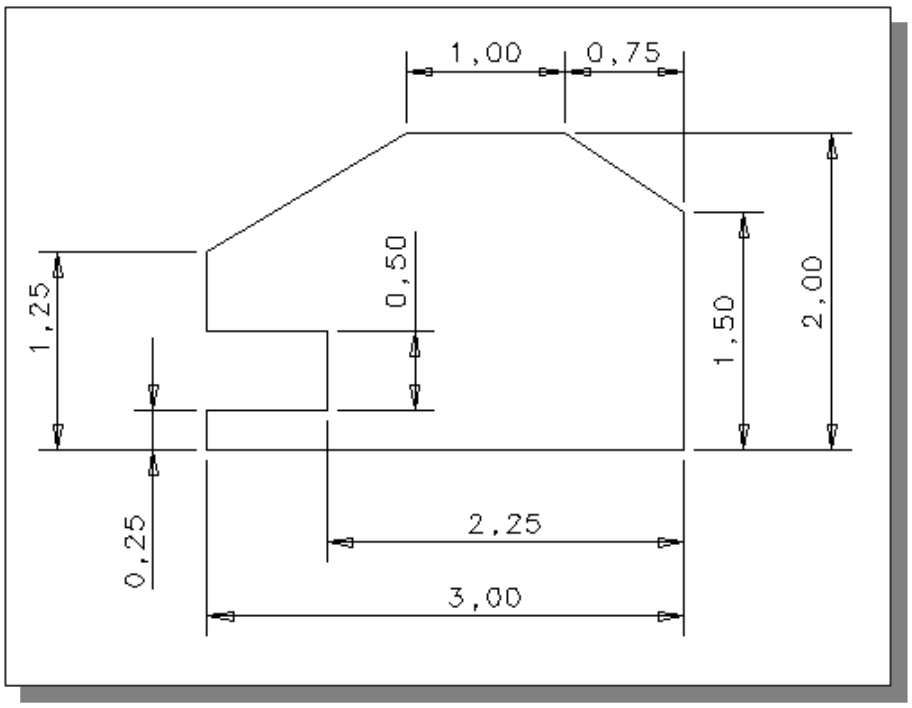


(d)

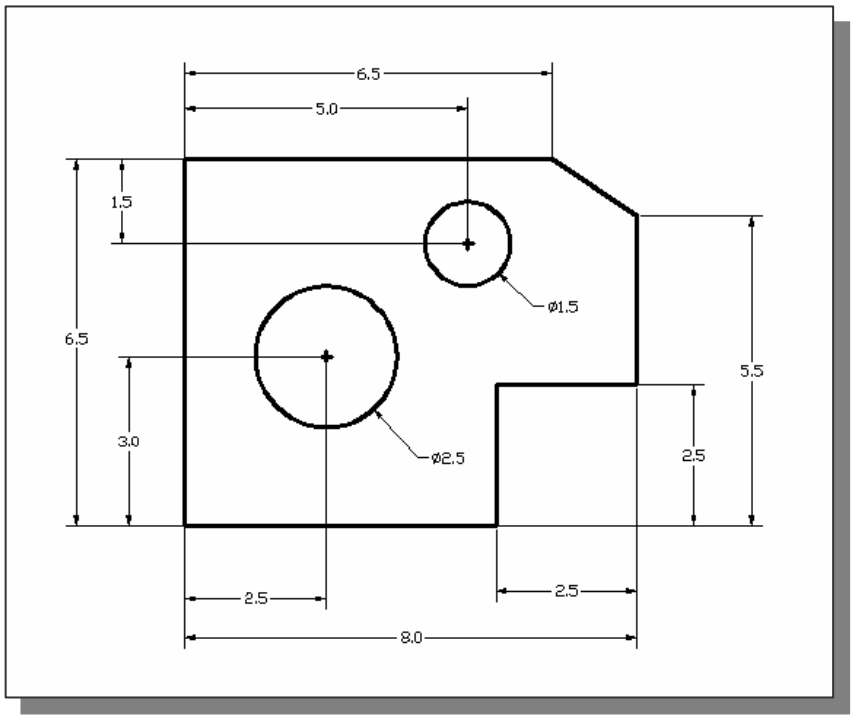


Exercises: (All dimensions are in inches.)

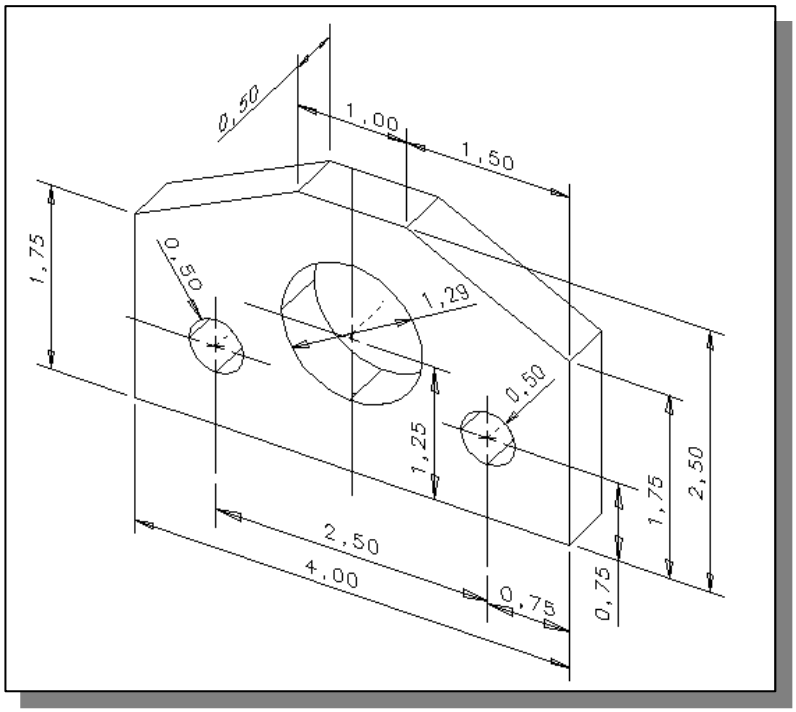
1. Plate Thickness: 0.25



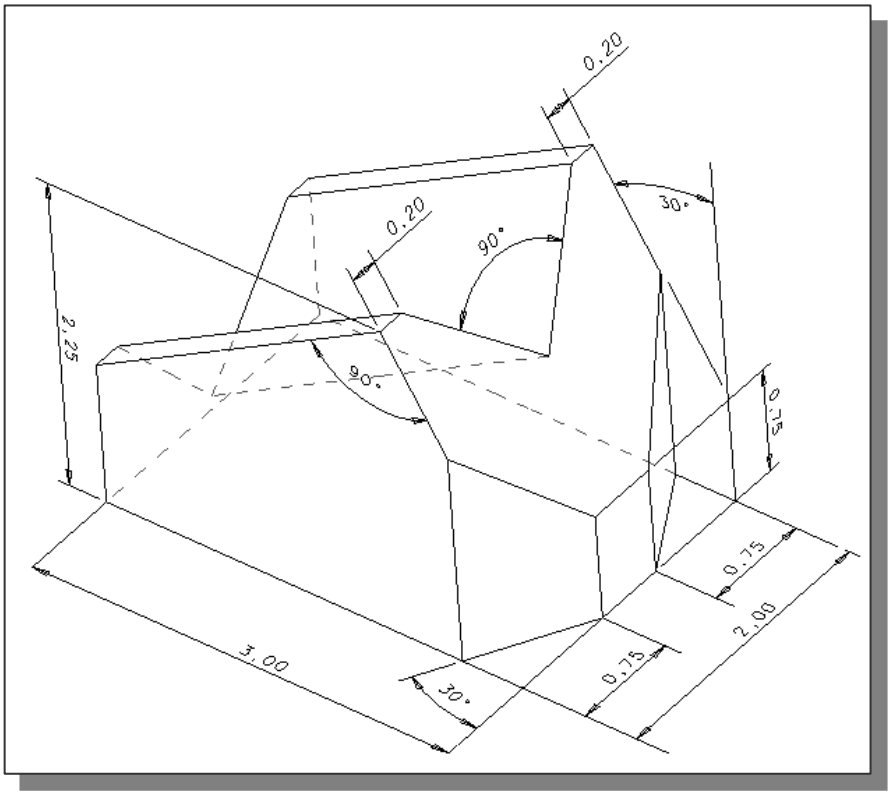
2.



3.


0,50
1,00
1,50
1,75
0,50
1,29
0,50
1,25
1,75
2,50
2,50
4,00
0,75
0,75

4.


0.20
0.20
90°
30°
2,25
90°
0,75
0,75
2,00
3,00
30°
0,75