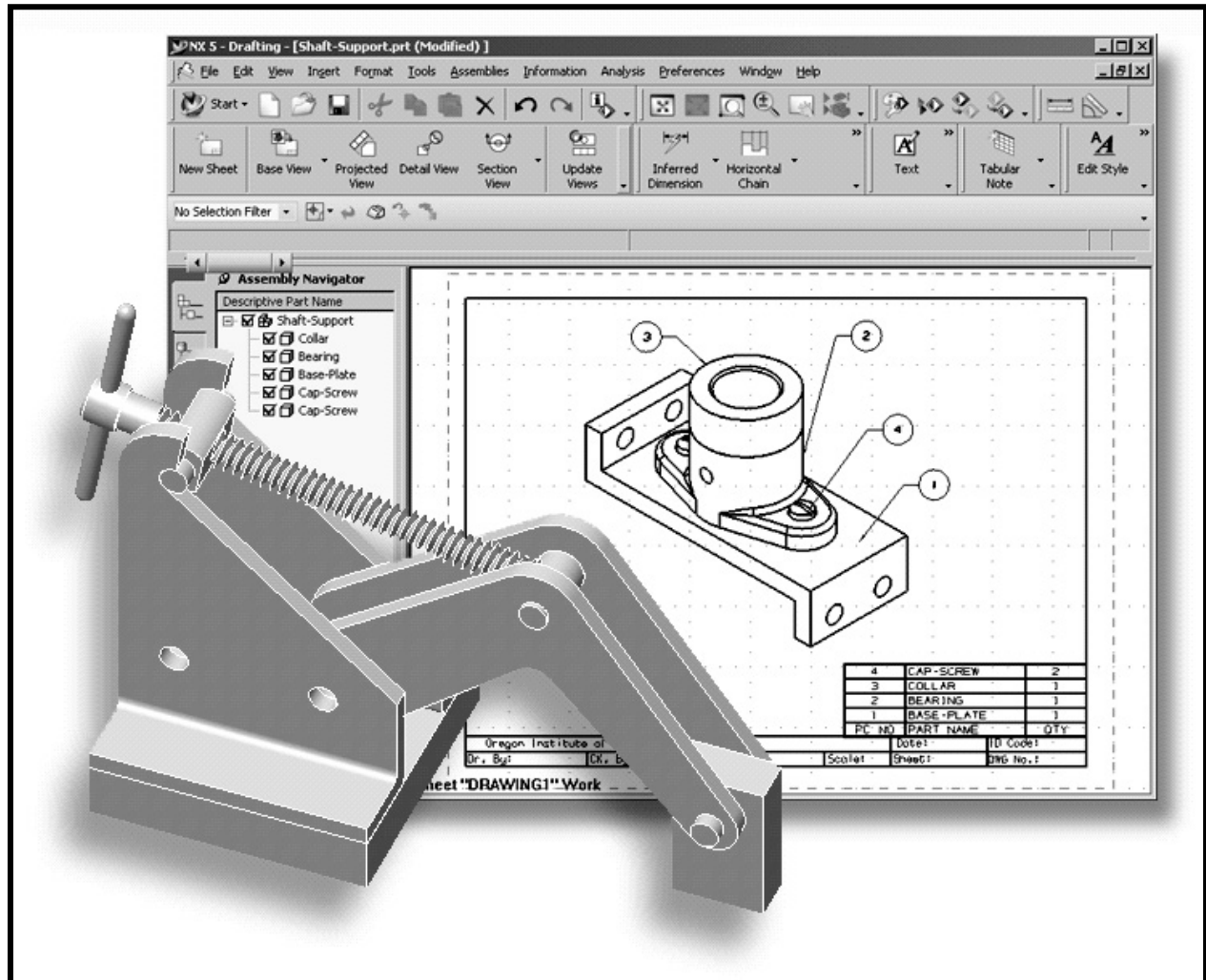


Parametric Modeling with UGS NX 5



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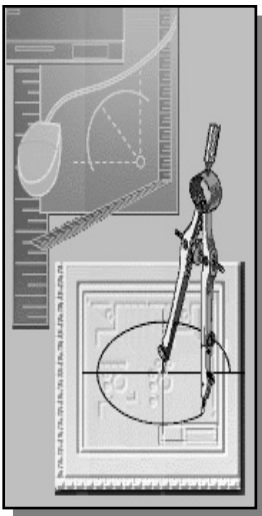
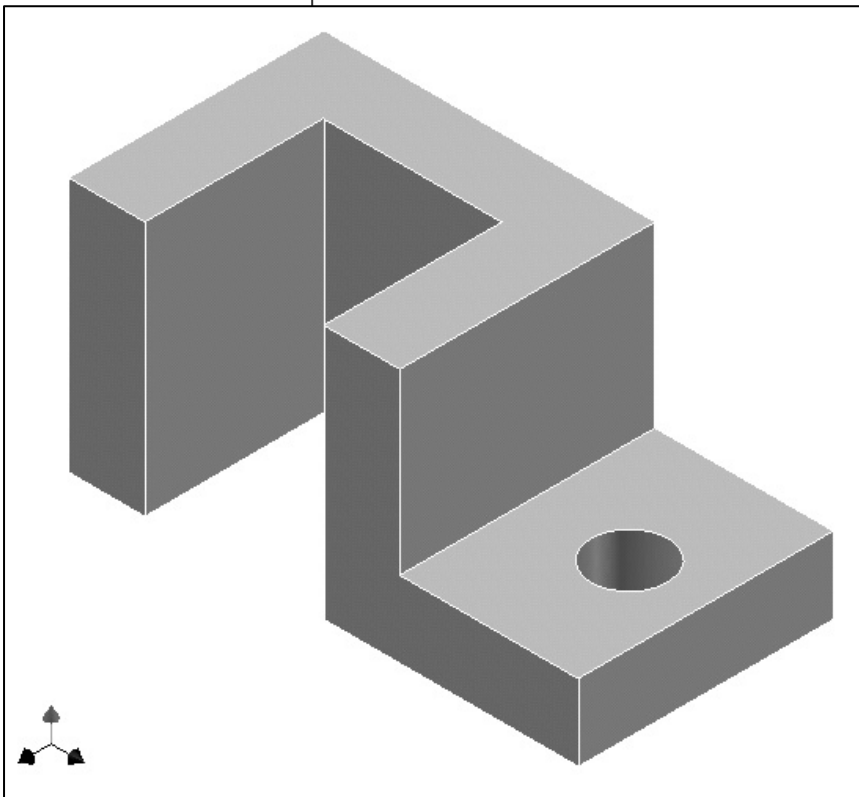
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Chapter 2

Parametric Modeling Fundamentals



Learning Objectives

- ◆ Create Simple Extruded Solid Models
- ◆ Understand the Basic Parametric Modeling Procedure
- ◆ Create 2-D Sketches
- ◆ Understand the "Shape before Size" Approach
- ◆ Use the Dynamic Viewing Commands
- ◆ Create and Edit Parametric Dimensions

Introduction

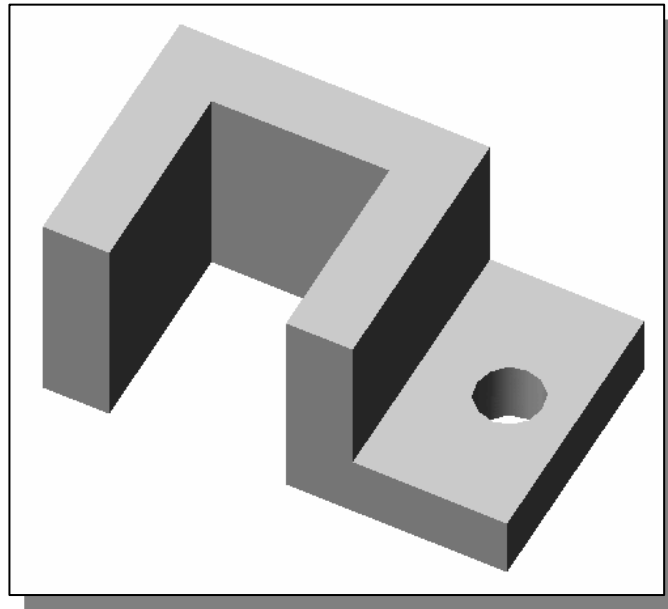
The **feature-based parametric modeling** technique enables the designer to incorporate the original **design intent** into the construction of the model. The word *parametric* means the geometric definitions of the design, such as dimensions, can be varied at any time in the design process. Parametric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

In *UGS NX*, the parametric part modeling process involves the following steps:

1. Set up **Units** and **Part name**.
2. Determine the type of the base feature, the first solid feature, of the design. Note that **Extrude, Revolve, or Sweep operations** are the most common types of base features.
3. Create a rough two-dimensional sketch of the basic shape of the base feature of the design.
4. Apply/modify constraints and dimensions to the two-dimensional sketch.
5. Transform the parametric two-dimensional sketch into a 3D solid.
6. Add additional parametric features by identifying feature relations and complete the design.
7. Perform analyses/simulations, such as finite element analysis (FEA) or cutter path generation (CNC), on the computer model and refine the design as needed.
8. Document the design by creating the desired 2D/3D drawings.

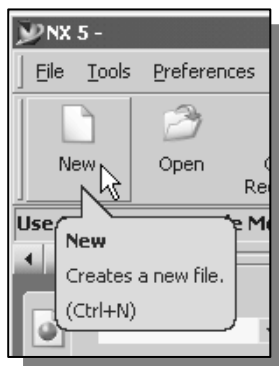
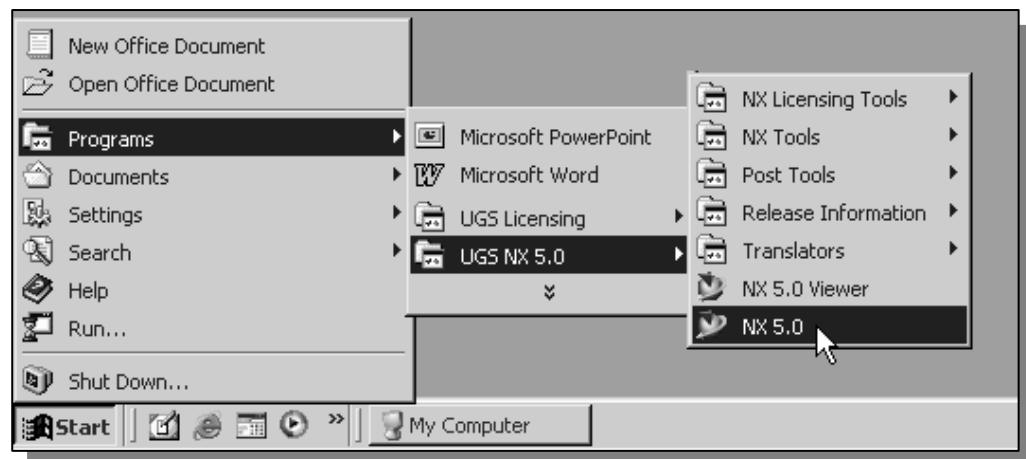
The approach of creating two-dimensional sketches of the three-dimensional features is an effective way to construct solid models. Many designs are in fact the same shape in one direction. Computer input and output devices we use today are largely two-dimensional in nature, which makes this modeling 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. *UGS NX* provides many powerful modeling and design-tools, and there are many different approaches to accomplishing 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 *UGS NX* user interface.

The *Adjuster* design



Step 1. Starting *UGS NX* and Units setup

1. Select the **UGS NX** option on the *Start* menu or select the **UGS NX** icon on the desktop to start *UGS NX*. The *UGS NX* main window will appear on the screen.

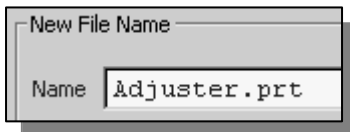
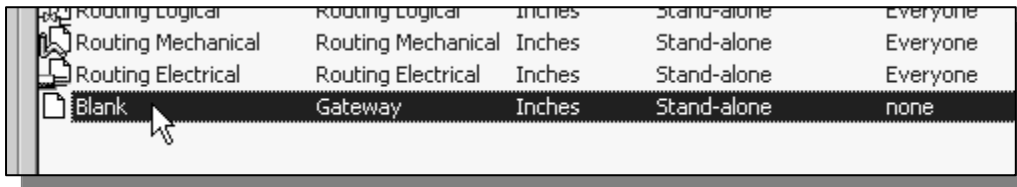


2. Select the **New** icon with a single click of the left-mouse-button (MB1) in the *Standard* toolbar area.
 - The *File New* dialog box will appear on the screen with three option tabs: **Model**, **Drawing** and **Simulation**. Note that the default is set to **Model** and we can create a new model using any of the pre-defined templates.



3. Select the **Inches** units as shown in the window. When starting a new CAD file, the first thing we should do is to choose the units we would like to use.

4. Select **Blank** in the *Templates* list. Note that the *Blank* template contains no pre-defined settings.



5. In the *New File Name* section, enter **Adjuster.prt** as the file *Name*.

6. Click **OK** to accept the settings and proceed with the **New File** command.



7. Select the **Start** icon with a single click of the left-mouse-button (MB1) in the *Standard* toolbar.

8. Pick **Modeling** in the pull-down list as shown in the figure.

- The above procedure shows the switching of the three ***NX* work areas**; we have gone from the “No Part” state to the “Applications” and “Task Environments” state. In the later chapters, the procedure to use the pre-defined template is also illustrated.

- Note that the *NX Modeling* application allows us to perform both the tasks of **parametric modeling** and **3D free-form surface modeling**.

Parametric Modeling

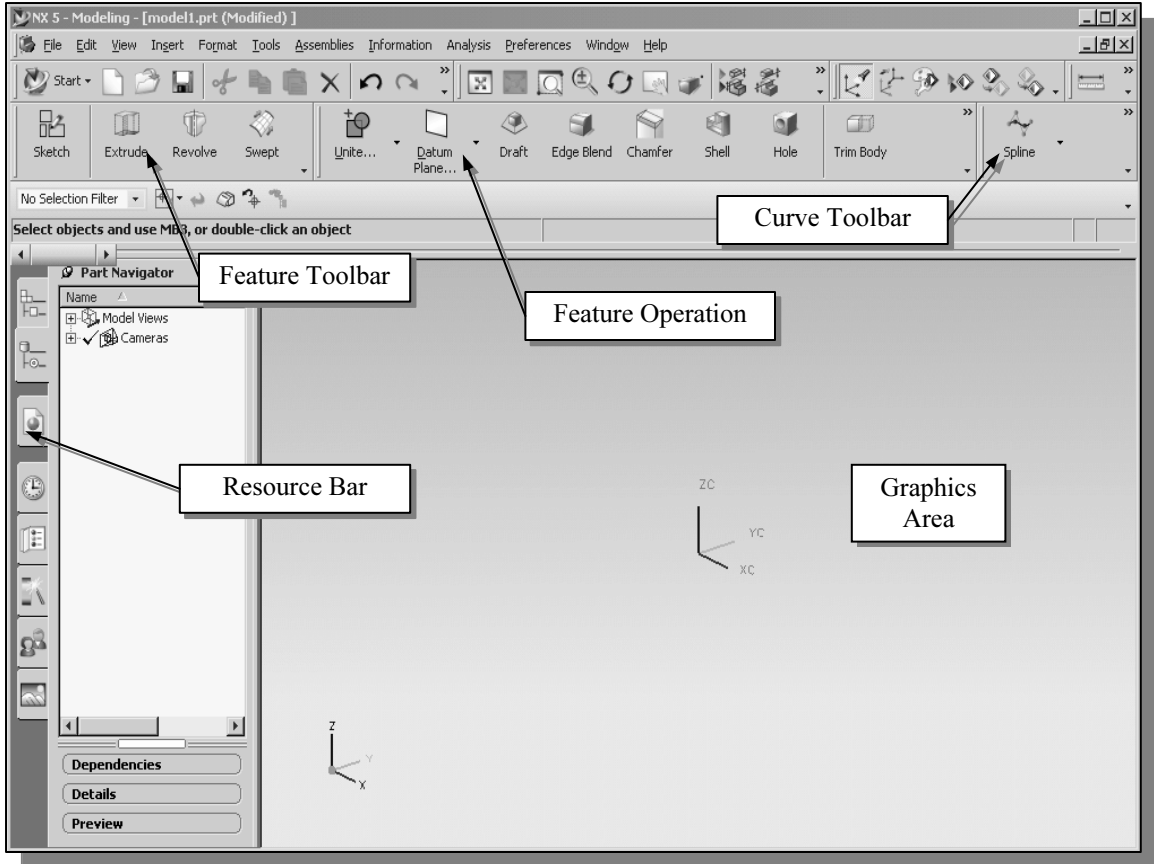
Under the *NX Modeling* application, a variety of tools are available for the creation of 2D and 3D wireframe models and associative feature-based solid models.

3D Free-Form Surface Modeling

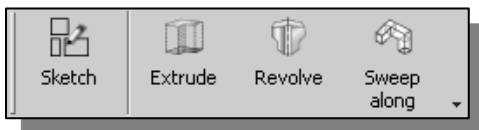
We can also perform more complex-shape modeling tasks, such as the creation of complex surface and solid models. For example, creating *swept features* along 3D curves; *lofted features* using conic methods; and *meshes* of points and curves.

UGS NX Application Screen Layout

- The *UGS NX Modeling* screen layout is quite similar to the *NX Gateway* application screen layout (refer to the figure in page 1-12). The new items on the screen include the *Form Feature* toolbar, the *Feature Operation* toolbar, and the *Curve* toolbar.

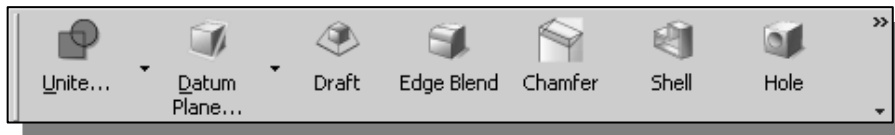


Form Feature Toolbar



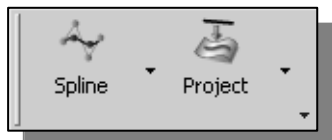
This toolbar contains tools that allow us to quickly create 2D sketches and form 3D features.

Feature Operation Toolbar



This toolbar contains tools that allow us to quickly create placed features.

Curve Toolbar



This toolbar contains tools that allow us to quickly access 2D curve options.

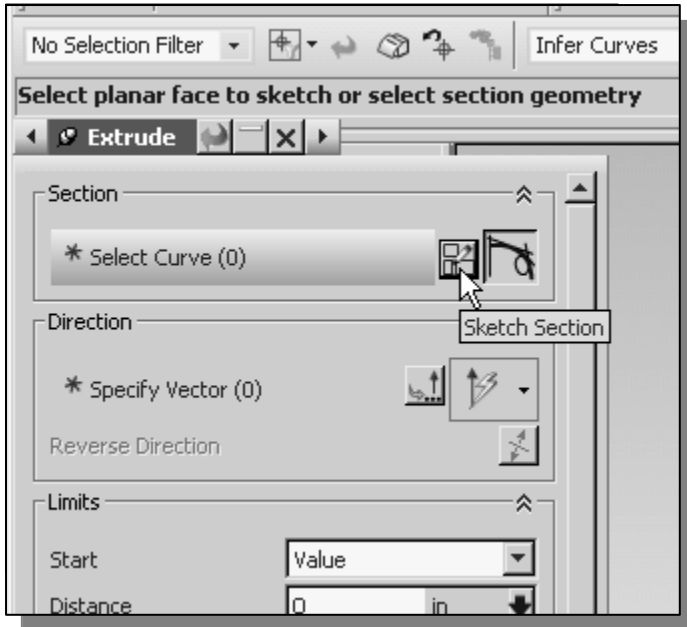
Step 2: Define/Set Up the First Solid Feature

- For the *Adjuster* design, we will create an extruded solid as the first feature.

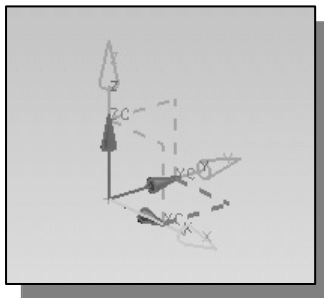


1. In the *Feature* toolbars (toolbars aligned to the right edge of the main window), select the **Extrude** icon as shown.

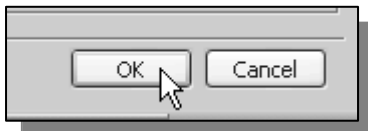
- The *Extrude* feature options dialog box, which contains applicable construction options, is displayed as shown in the figure below.



2. On your own, move the cursor over the icons and read the brief descriptions of the different options available. Note that the default *Extrude* option is set to **Select Section**.
3. Click the **Select Section** button to begin creating a new *2D sketch*.

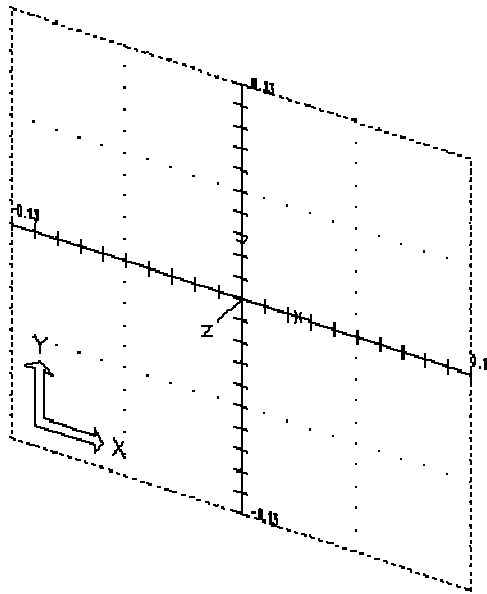


- ❖ Note the default *sketch plane* is aligned to the **XC-YC plane** of the displayed *work coordinate system*.



4. Click **OK** to accept the default setting of the *sketch plane*.

Work 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 its 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 the **absolute 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 absolute space. The absolute space is usually a **3D Cartesian coordinate system** that the user cannot change or manipulate.

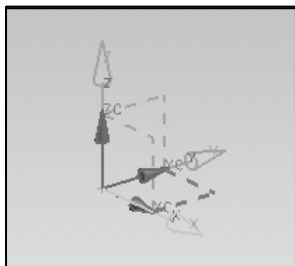
In most engineering designs, models can be very complex, and it would be tedious and confusing if only one coordinate system, the absolute coordinate system, was available. Practical 3D modeling systems allow the user to define **Work Coordinate Systems (WCS)**, also known as **Local Coordinate Systems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

The basic concepts of coordinate systems remain the same for most CAD systems, but the actual usage of a particular type of coordinate system may vary greatly from one CAD system to another. *UGS NX* has two primary coordinate systems: the **Absolute Coordinate System (ACS)** and the **Work Coordinate System (WCS)**. The ACS is fixed in space, whereas the WCS is a mobile system to facilitate geometry construction in different orientations. The WCS can be located and oriented anywhere in model space.

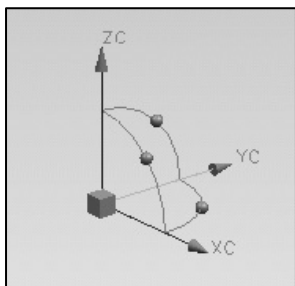
Although 3D objects are generally created and stored in 3D space coordinates, most of the 3D geometry entities can be referenced using 2D Cartesian coordinate systems. Typical input devices such as a mouse or digitizers are two-dimensional by nature; the movement of the input device is interpreted by 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 *UGS NX sketch plane* is a special construction tool that enables the planar nature of 2D input devices to be directly mapped into the 3D coordinate systems. The *sketch plane* is part of the *NX* work coordinate system that can be aligned to the absolute coordinate system, an existing face of a part, or a reference plane. By default, the *sketch plane* is aligned to the *XY* plane of the default *WCS*; note that the default *WCS* is aligned to the absolute coordinate system (*ACS*).

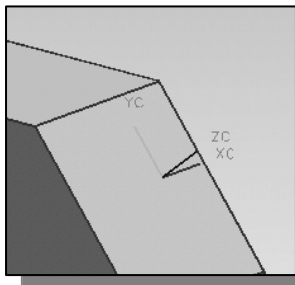
Think of a sketch plane as the surface on which we can sketch the 2D sections of the parts. It is similar to a piece of paper, a white board, or a chalkboard that can be attached to any planar surface. The first profile we create is usually drawn on one of the work planes of the work coordinate system (*WCS*). Subsequent profiles can then be drawn on sketch planes that are defined on **planar faces of a part, work planes aligned to part geometry, or work planes attached to a coordinate system** (such as the *XY*, *XZ*, and *YZ* planes of *WCS*).



- ❖ The default *sketch plane* is aligned to the *XY* plane of the work coordinate system.



- ❖ The *work coordinate system* (*WCS*) is a mobile system to assist geometric construction in different orientations. The *WCS* can be located and oriented anywhere in model space.



- ❖ Note that most solid modeling operations do not require manipulation of the *WCS*. Features are generally added relative to existing geometry of the solid model.

Creating Rough Sketches

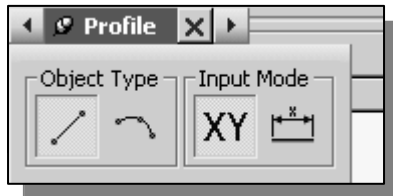
Quite often during the early design stage, the shape of a design may not have any precise dimensions. Most conventional CAD systems require the user to input the precise lengths and locations of all geometric entities defining the design, which are not available during the early design stage. With *parametric modeling*, we can use the computer to elaborate and formulate the design idea further during the initial design stage. With *UGS NX*, 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, a **rough sketch** is not precise at all. When sketching, we simply sketch the geometry so that it closely resembles the desired shape. Precise scale or lengths are not needed. *UGS NX* 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 *UGS NX*:

- **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, start by creating an angle that is 50 or 60 degrees (and make adjustment later). Otherwise, *UGS NX* might assume the intended angle to be a 90-degree angle.
 - **Draw the geometry so that it does not overlap.** To create a 3D features from a 2D sketch, the 2D geometry used should define a clear boundary. *Self-intersecting* geometry shapes are not allowed, as it cannot be converted into a solid feature.
 - **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, to those of conventional computer aided drafting. In order to understand and fully utilize *UGS NX*'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 Computer Aided Drafting systems.

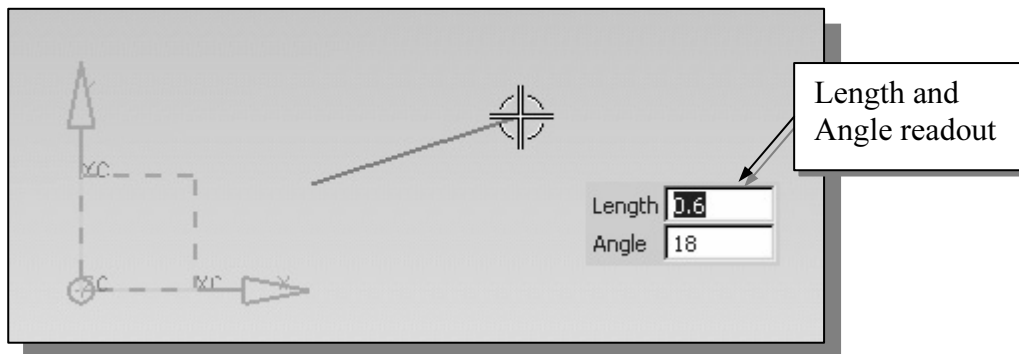
Step 3: Creating a Rough 2D Sketch

- The *Sketch Curve* toolbar provides tools for creating and editing of the basic 2D geometry, construction tools such as lines and circles, and editing tools such as trim and extend. Notice, by default, the **Profile** tool is activated. The **Profile** tool allows us to create lines and/or arcs that are joined together.

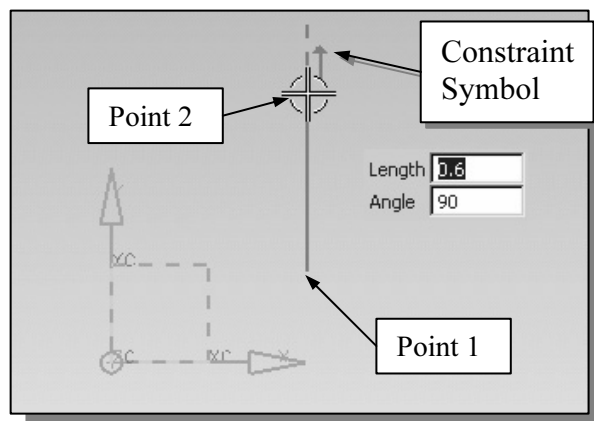


- Notice, by default, the **Line** option is activated. The **Line** option allows us to create line segments that are joined together.

1. As you move the cursor, you will see a digital readout next to the cursor. The readout gives you the cursor location relative to the Work Coordinate System. Pick a location that is toward the right side of the WCS as the first point of a line.
2. Move the cursor around and you will notice the readout, next to the cursor, providing the length and angle of the line.















3. Move the graphics cursor directly above the previous point and create a vertical line as shown below (**Point 2**). Notice the geometric constraint symbol displayed.

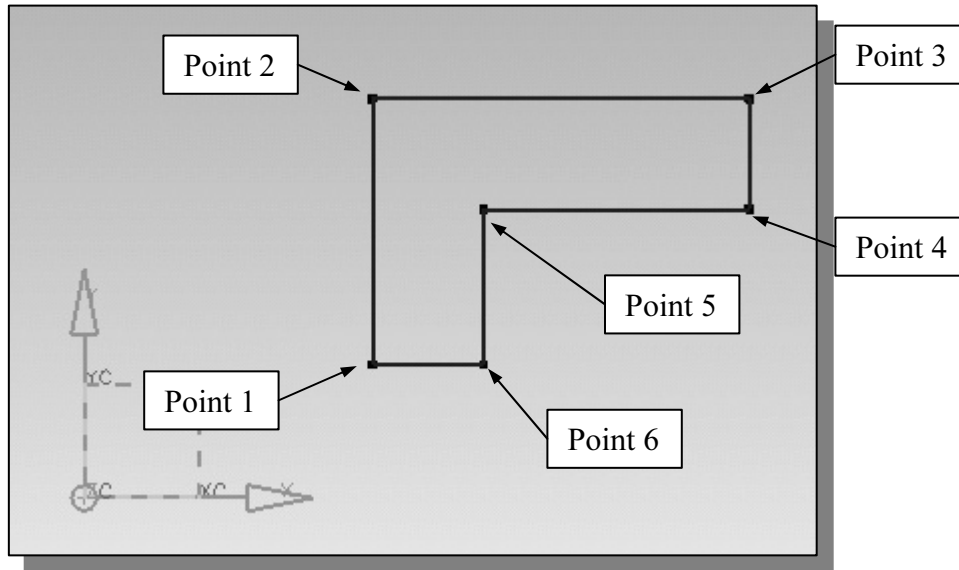


Geometric Constraint Symbols

UGS NX displays different visual clues, or symbols, to show you alignments, perpendicularities, tangencies, etc. These constraints are used to capture the *design intent* by creating constraints where they are recognized. *UGS NX* displays the governing geometric rules as models are built. To prevent constraints from forming, hold down the [Alt] key while creating an individual sketch curve. For example, while sketching line segments with the Line command, endpoints are joined with a **Coincident** constraint, but when the [Alt] key is pressed and held, the inferred constraint will not be created.

	Vertical	indicates a line is vertical
	Horizontal	indicates a line is horizontal
	Dashed line	indicates the alignment is 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
	Coincident	indicates the cursor is at the endpoint of an entity
	Concentric	indicates the cursor is at the center of an entity
	Tangent	indicates the cursor is at tangency points to curves
	Midpoint	indicates the cursor is at the midpoint of an entity
	Point on Curve	indicates the cursor is on curves
	Equal Length	indicates the length of a line is equal to another line
	Equal Radius	indicates the radius of an arc is equal to another arc

- Complete the sketch as shown below, creating a closed region ending at the starting point (Point 1). Do not be overly concerned with the actual size of the sketch. Note that all line segments are sketched horizontally or vertically.

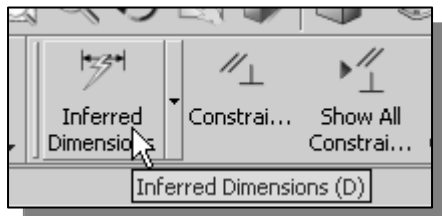
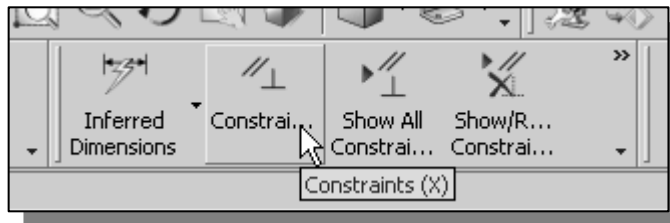


- Inside the graphics window, click twice with the **middle-mouse-button (MB2)**, or press the **[Esc]** key once, to end the **Sketch Line** command.

Step 4: Apply/Modify Constraints and Dimensions

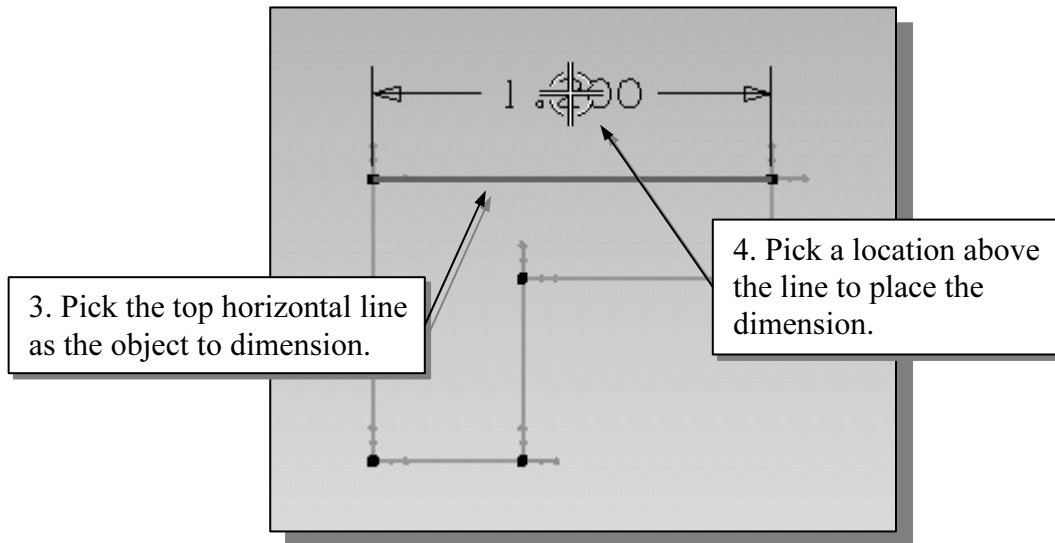
- As the sketch is made, *UGS NX* automatically applies some of the geometric constraints (such as **Horizontal**, **Parallel**, and **Perpendicular**) to the sketched geometry. We can continue to modify the geometry, apply additional constraints, and/or define the size of the existing geometry. In this example, we will illustrate adding dimensions to describe the sketched entities.

- Move the cursor to the **Sketch Constraints** toolbar area, which is located to the right of the **Sketch Curve** toolbar.

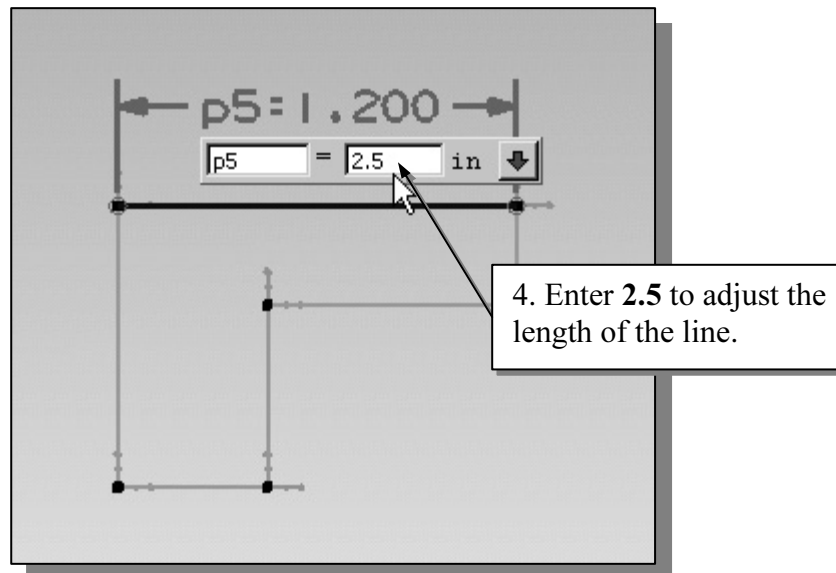


- Activate the **Inferred Dimensions** command by left-clicking once on the icon as shown. The **Inferred Dimensions** command allows us to quickly create and modify dimensions.

3. The message “*Select object to Dimension or select dimension to edit*” is displayed in the *Message* area. 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. (Note that the value displayed on your screen might be different than what is shown in the figure above.)
- ❖ The **Inferred Dimension** command will create a length dimension if a single line is selected. The current length of the line is displayed, in the **edit box**, next to the dimension as shown in the below figure.
5. Enter **2.5** as the desired length of the line. (Press the **ENTER** key once after you enter the new value.)



- *UGS NX* will now update the profile with the new dimension value. Note that in parametric modeling, dimensions are used as **control variables**.

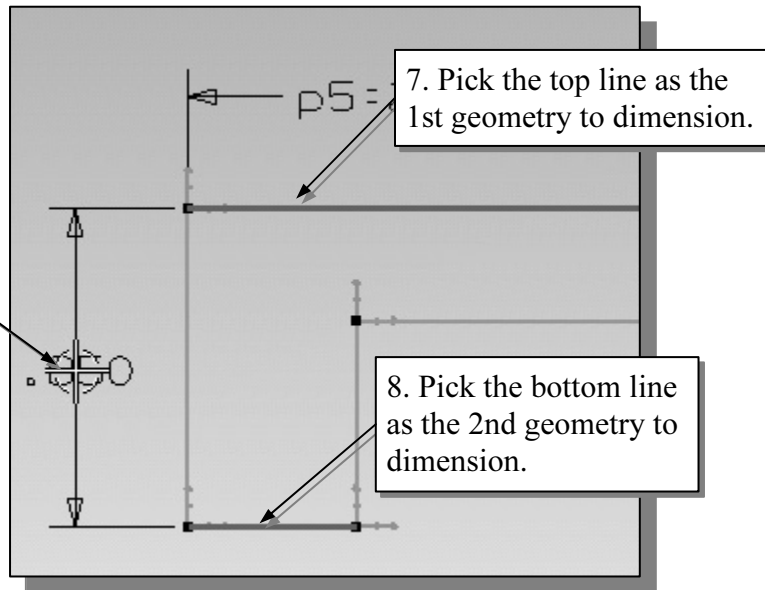


6. Click the **Fit** button to resize the display window.

❖ Note that we are returned to the Inferred Dimensions command; the **Fit** command is executed without interrupting the current active command.

7. The message “*Select object to Dimension or select dimension to edit*” is displayed in the *Message* area. Select the top horizontal line as shown below.

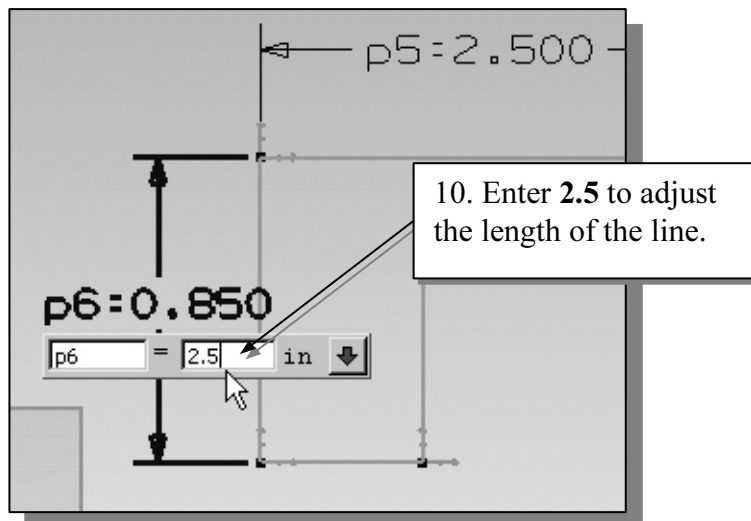
8. Select the bottom horizontal line as shown below.



9. Place the dimension next to the sketch.

9. Pick a location to the left of the sketch to place the dimension.

10. Enter **2.5** as the desired length of the line.



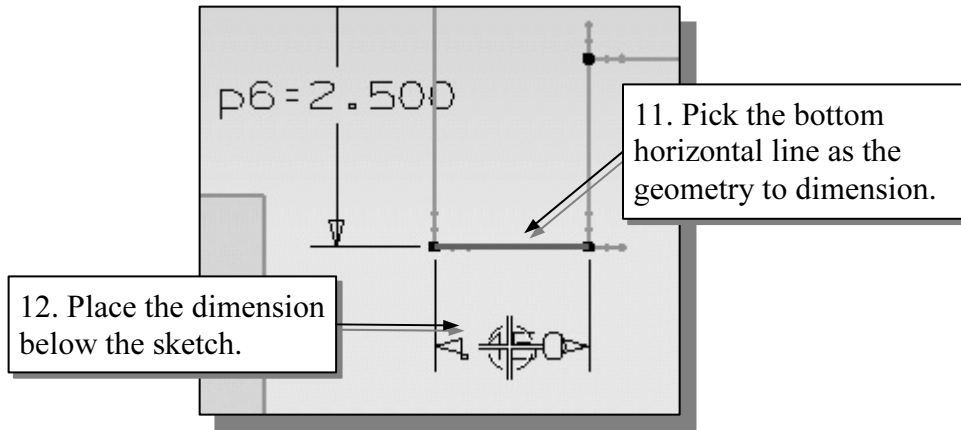
❖ When two parallel lines are selected, the Inferred Dimension command will create a dimension measuring the distance between them.



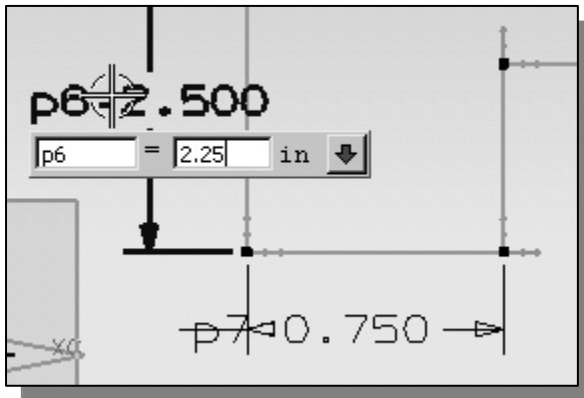
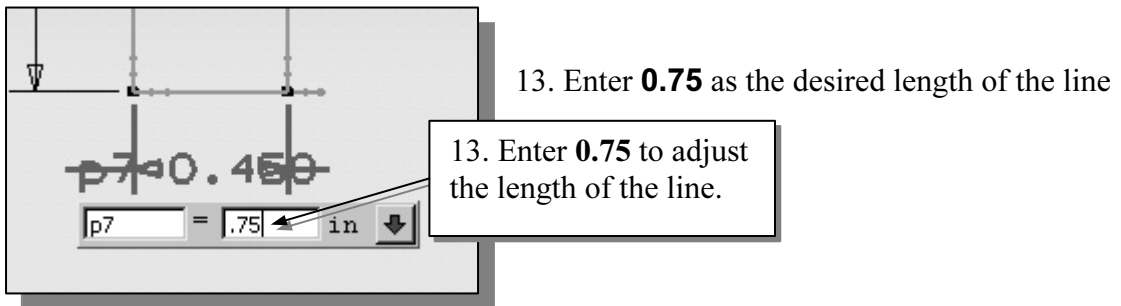
❖ **Quick Zoom function by turning the mouse wheel**

Turning the **mouse wheel** can adjust the scale of the display. Turning forward will reduce the scale of the display, using the cursor location as the center reference, making the entities display smaller on the screen. Turning backward will magnify the scale of the display.

11. The message “*Select object to dimension or select dimension to edit*” is displayed in the *Message* area. Select the bottom horizontal line as shown below.



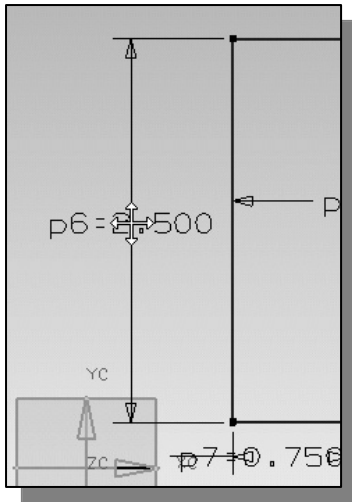
12. Pick a location below the sketch to place the dimension.



14. Pick the vertical dimension and notice the *Inferred Dimensions* command can also be used to adjust any existing dimensions.

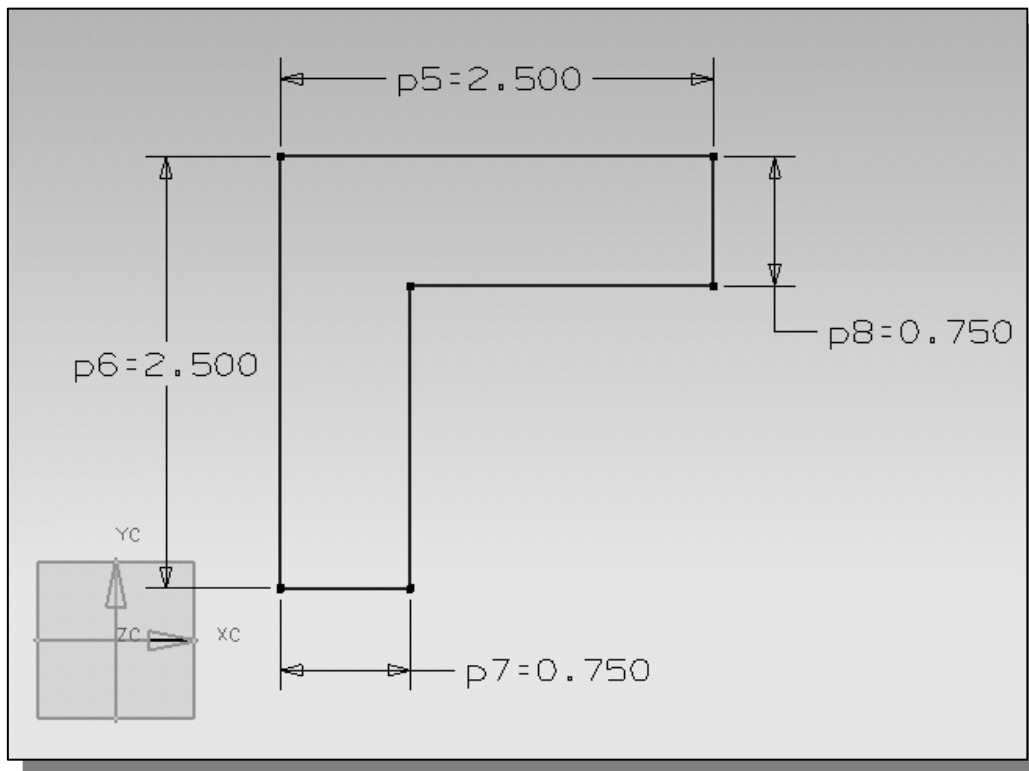
15. On your own, enter any number and observe the adjustment done by *UGS NX*.

16. Inside the graphics window, click twice with the **middle-mouse-button (MB2)**, or hit the **[Esc]** key once, to end the Inferred Dimensions command.



17. To reposition any of the dimensions, simply **press and drag** with the **left-mouse-button (MB1)** on the dimension as shown in the figure.

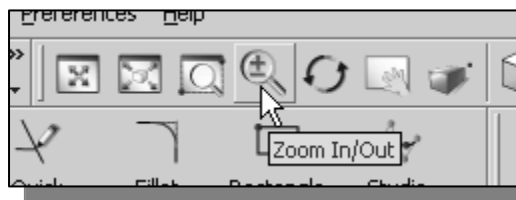
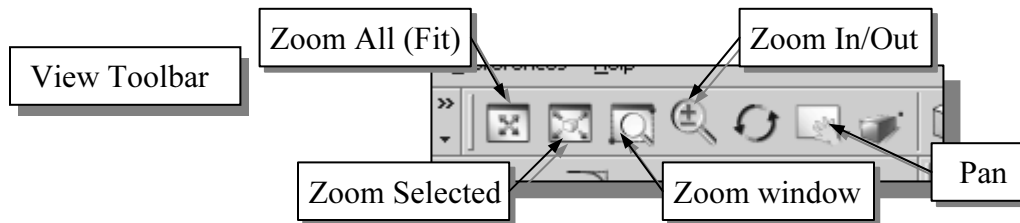
18. On you own, repeat the above steps, create and modify dimensions so that the sketch appears as shown in the figure below.



- ❖ The **Inferred Dimensions** command is also known as the **Smart Dimension** command; dimensions are created based on the selection of the types and orientations of objects. You might want to read through this section again and consider the powerful implication of this command. Also consider this question: How would you create an *angle dimension* using the Inferred Dimensions command?

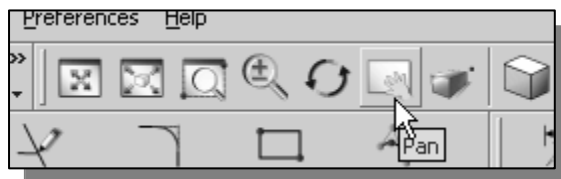
Dynamic Viewing Functions – *Zoom* and *Pan*

- *UGS NX* provides a special user interface called *Dynamic Viewing* that enables convenient viewing of the entities in the graphics window.



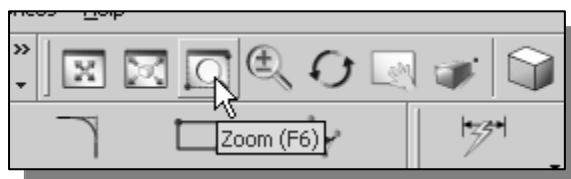
1. Click on the **Zoom In/Out** icon, located in the *Standard* toolbar as shown.

2. Move the cursor near the center of the graphics window.
3. Inside the graphics window, **press and drag with the left-mouse-button**, then move **upward** to enlarge the current display scale factor.
4. Inside the graphics window, **press and drag with the left-mouse-button**, then move **downward** to reduce the current display scale factor.
5. Click once with the **middle-mouse-button (MB2)**, or press the [**Esc**] key once, to exit the **Zoom** command.



6. Click on the **Pan** icon, located in the *View* toolbar area. The icon is the picture of a hand.

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



7. Click on the **Zoom Window** icon, located in the *View* toolbar area. (Function key [**F6**] can also be used.)

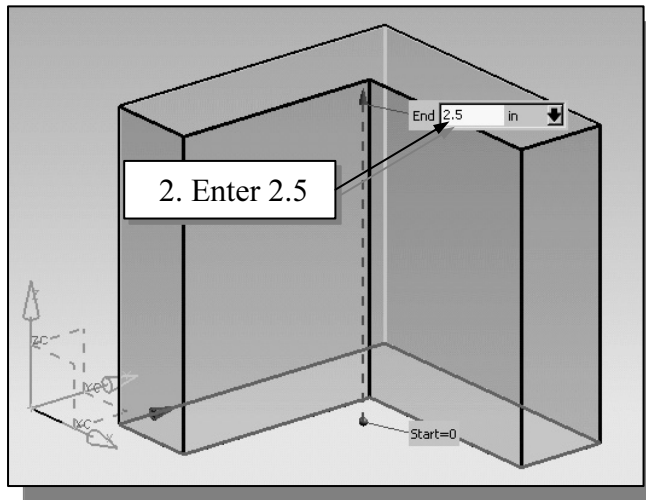
8. Select two points, forming the two opposite corners of a rectangle, to define the **Zoom window** region.
9. The current display is resized/mapped to the selected region.
10. On your own, use the available **Zoom** and **Pan** options to reposition the sketch near the center of the screen.

Step 5: 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 section. Extruding a 2D section 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 *UGS NX*, 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*.



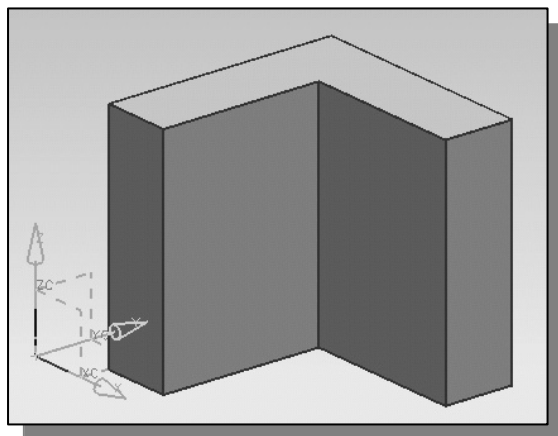
1. Select **Finish Sketch** by clicking once with the left-mouse-button (MB1) on the icon.



2. In the *Extrude* popup window, enter **2.5** as the extrusion *Distance*. Notice that the sketch region is automatically selected as the extrusion section.



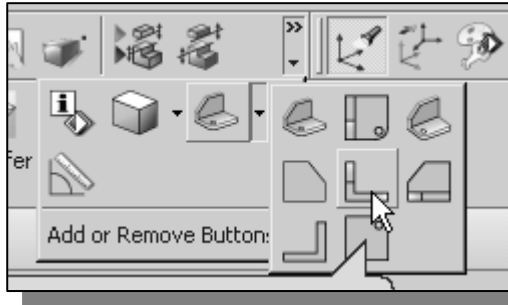
3. Click on the **OK** button to proceed with creating the 3D part.



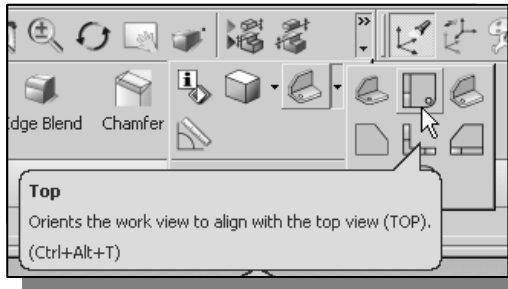
- Note that all dimensions disappeared from the screen. All parametric definitions are stored in the *UGS NX database* and any of the parametric definitions can be displayed and edited at any time.

Display Orientations

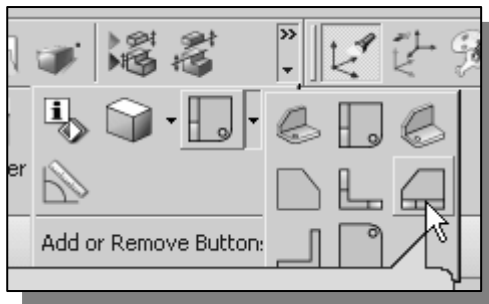
- ❖ *UGS NX* provides many ways to display views of the three-dimensional design. Several options are available that allow us to quickly view the design to track the overall effect of any changes being made to the model. We will first orient the model to display in the *front view*, by using the *View* toolbar.



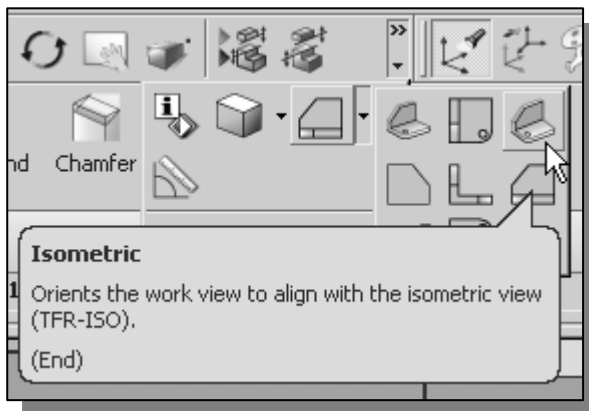
1. Select **Front View** in the *View* toolbar to change the display to the front view. (Note that **[Ctrl+Alt+F]** can also be used to activate this command.)



2. Select **Top View** in the *View* toolbar to change the display to the top view. (Note that the **[Ctrl+Alt+T]** key can also be used to activate this command.)



3. Select **Right View** in the *View* toolbar to change the display to the right side view. (Note that the **[Ctrl+Alt+R]** key can also be used to activate this command.)



4. Select **Isometric View** in the *View* pull-down menu to change the display to the isometric view. (Note that the **[End]** key can also be used to activate this command.)

- ❖ Notice the other view-related commands that are available under the *View* toolbar. Most of these commands can be accessible through the toolbar and/or *function keys*.

Dynamic Viewing – Icons, Mouse buttons and Quick keys

- The **3D Rotate** icon is the icon with two circular arrows in the *View* toolbar. *3D Rotate* enables us to manipulate the view of 3D objects by clicking and dragging with the left-mouse-button:



1. Click on the **Rotate** icon in the *View* toolbar.

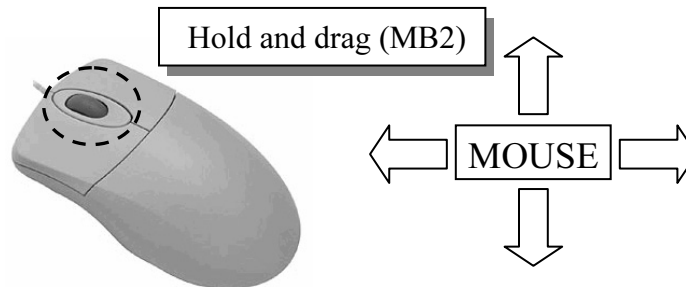
2. Drag with the left-mouse-button (MB1) for free rotation of the 3D model.
3. Press the [F7] key (toggle switch) once to exit the 3D Rotate command.

- We can also use the mouse buttons to access the *Dynamic Viewing* functions.

❖ 3D Dynamic Rotation – The Middle-Mouse-Button (MB2)

Hold and drag with the middle-mouse-button to rotate the display interactively.

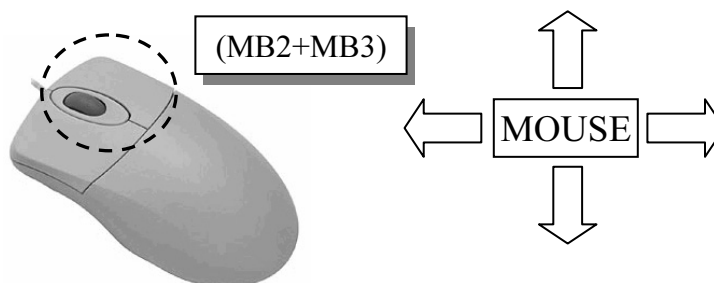
Dynamic Rotation



❖ Panning – (1) The Middle and Right Mouse Buttons

Hold and drag with the middle and right mouse buttons (MB2+MB3) to pan the display. This allows you to reposition the display while maintaining the same scale factor of the display.

Pan

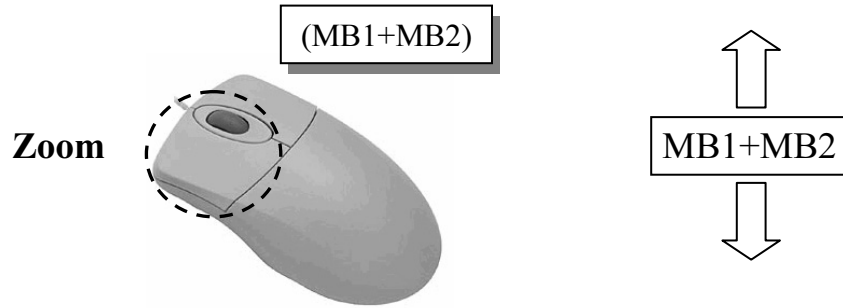


(2) [Shift] + [Middle-mouse-button (MB2)]

Hold the [SHIFT] key and the **middle-mouse-button (MB2)** to Pan the display.

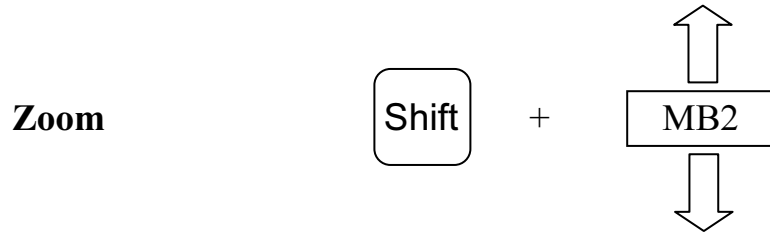
❖ **Zooming – The Left and Middle Mouse Buttons**

Hold and drag with the left and middle mouse buttons (MB1+MB2) vertically on the screen to adjust the scale of the display. Moving upward will magnify the scale of the display, making the entities display larger on the screen. Moving downward will reduce the scale of the display.



(2) [Shift] + [Middle mouse button (MB2)]

Hold the [SHIFT] key and the **middle-mouse-button (MB2)** to pan the display.

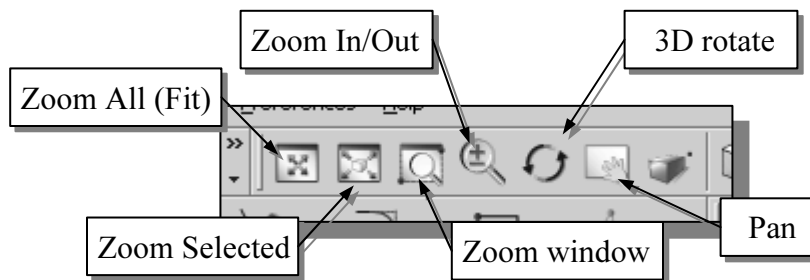


(3) Turning the mouse wheel

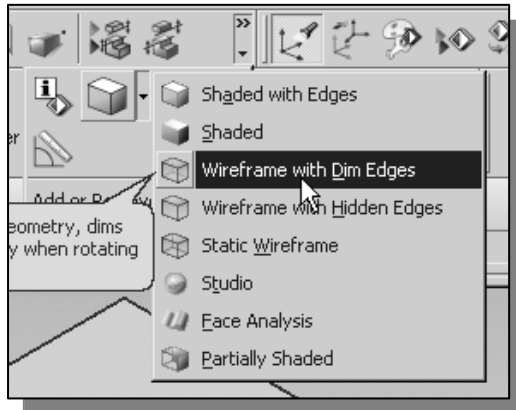


Turning the mouse wheel can also adjust the scale of the display. Turning forward will reduce the scale of the display, making the entities display smaller on the screen. Turning backward will magnify the scale of the display.

➤ Note that the *dynamic viewing* functions are also available through the **View** toolbar.

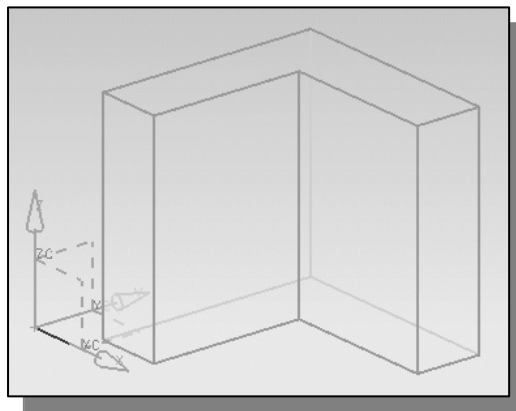


Display Modes



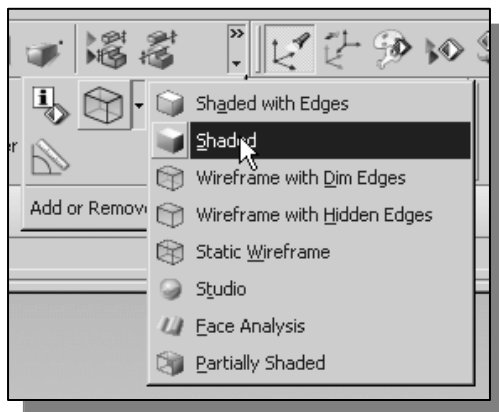
- The display in the graphics window has two basic display modes: wireframe and shaded display. To change the display mode in the active window, click on the triangle icon next to the display mode button on the *View* toolbar, as shown in the figure.

- Pick the **Wireframe with Dim Edges** option as shown in the figure.



❖ Wireframe with Dim Edges

This display mode generates an image of the 3D object with all the back lines shown as lighter edges.

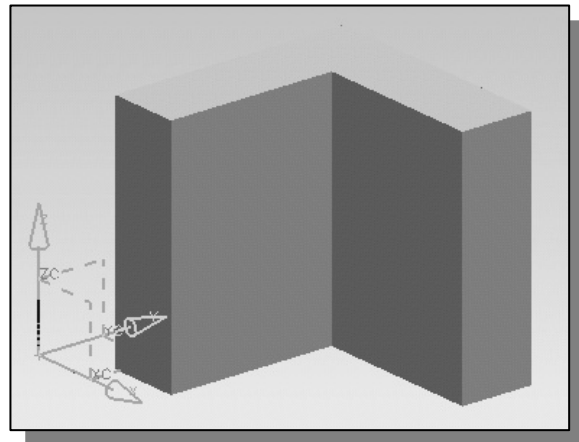


- Pick the **Shaded** option as shown.

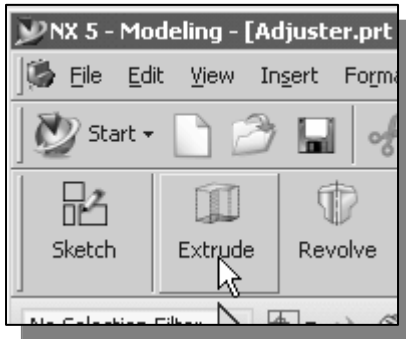
❖ Shaded Image

This display mode option generates a shaded image of the 3D object without the edges highlighted.

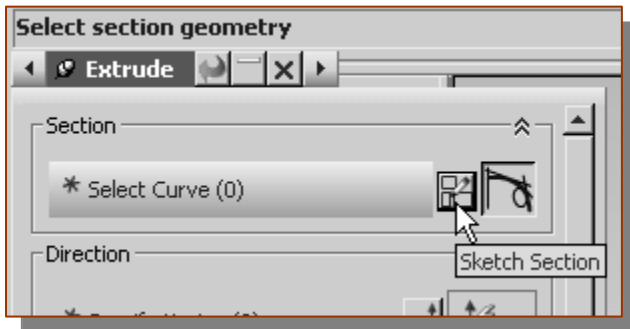
- On your own, use the different options to familiarize yourself with the 3D viewing/display commands.



Step 6-1: Adding an Extruded Feature

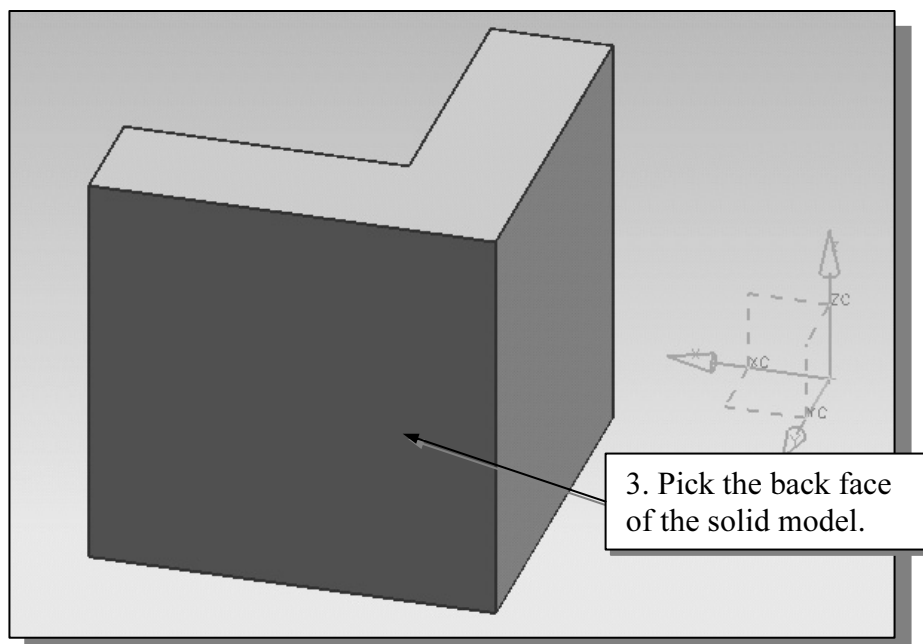


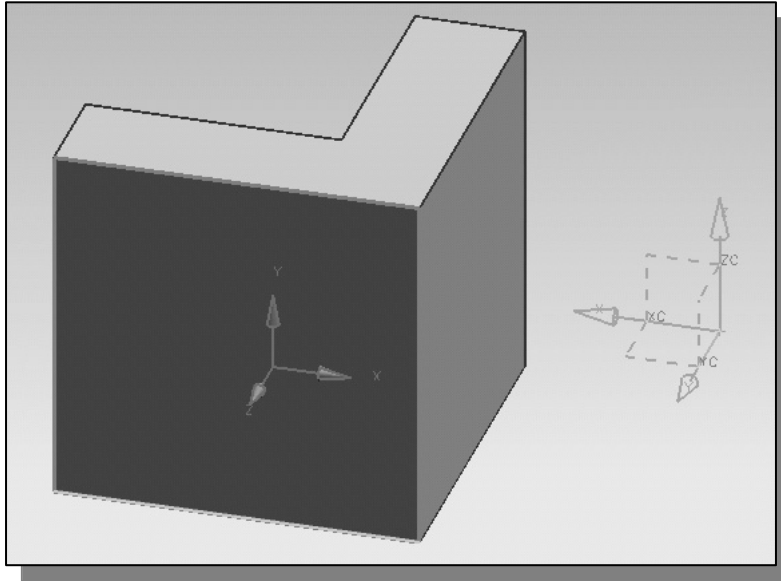
1. In the *Feature* toolbars (toolbars aligned to the right edge of the main window), select the **Extrude** icon as shown.



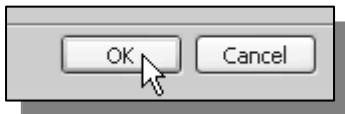
2. Click the **Sketch Section** button to begin creating a new *2D sketch*.

3. In the *Message* area, the message: “*Select planar face to sketch or select section geometry*” is displayed. *UGS NX* expects us to identify a planar surface where the 2D sketch of the next feature is to be created. Move the graphics cursor on the 3D part and notice that *UGS NX* will automatically highlight feasible planes and surfaces as the cursor is on top of the different surfaces. Pick the back vertical face of the 3D solid object, parallel to the **XZ plane**, as shown.



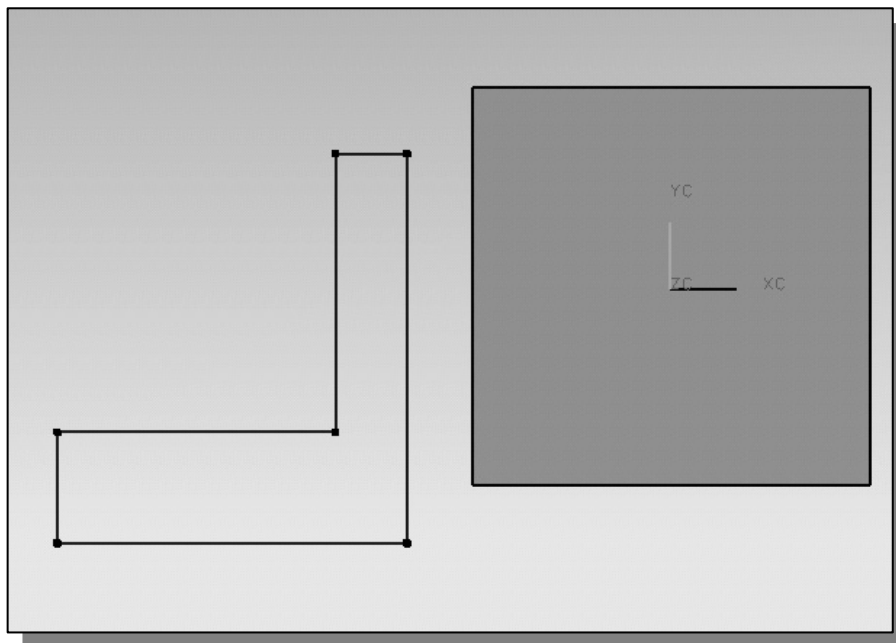


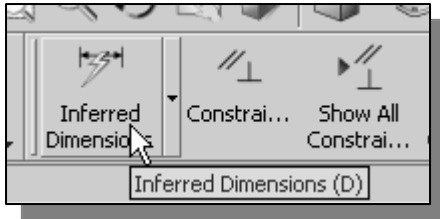
- Note that the *sketch plane* is aligned to the selected face. *UGS NX* automatically establishes a Work Coordinate System (WCS), and records its location with respect to the part on which it was created.



4. Click **OK** to accept the selected face and proceed with the new WCS.

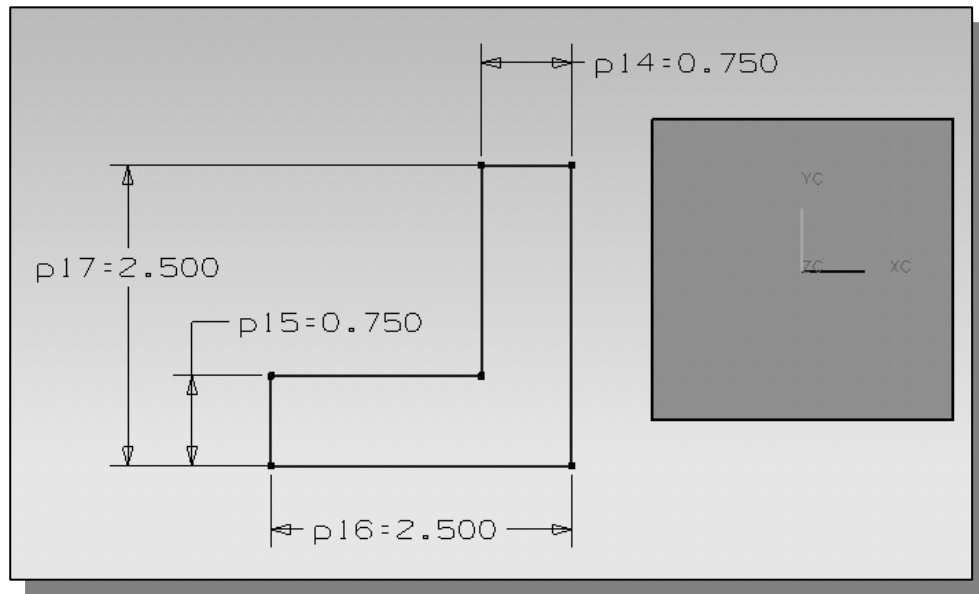
- Next, we will create another 2D sketch, which will be used to create another extrusion feature that will be added to the existing solid object.
5. On your own, create a 2D sketch consisting of horizontal/vertical lines as shown in the figure below. Note that we intentionally do not want to align any of the corners to the 3D model.



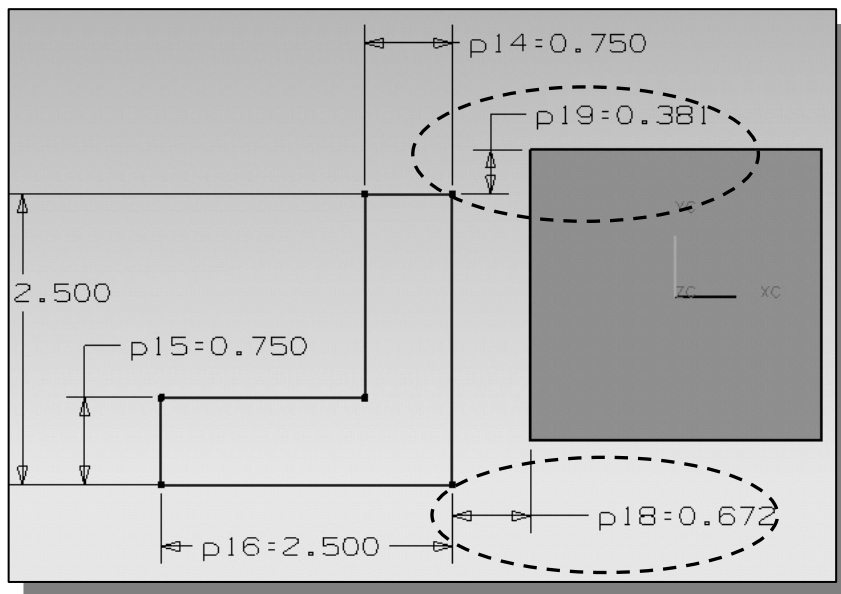


6. Activate the **Inferred Dimensions** command by left-clicking once on the icon as shown. The Inferred Dimensions command allows us to quickly create and modify dimensions

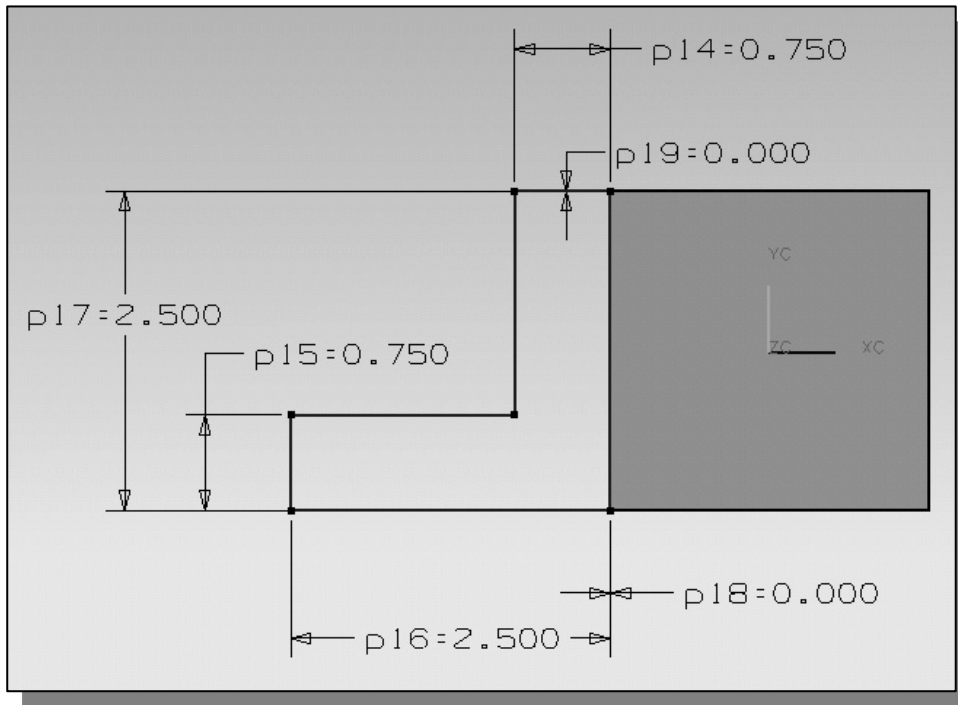
7. The message “*Select object to dimension or select dimension to edit*” is displayed in the *Message* area, at the bottom of the *UGS NX* window. Create and modify the four dimensions to describe the size of the sketch as shown in the figure.



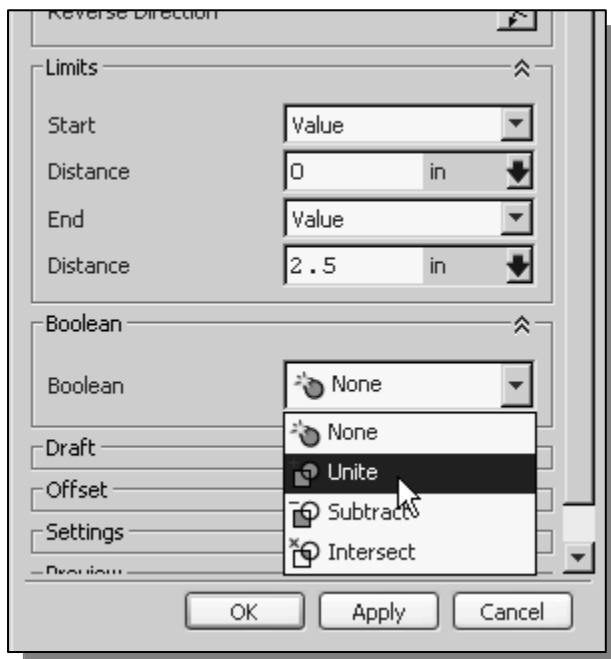
8. Create the two location dimensions to describe the position of the sketch relative to the top corner of the solid model as shown.



9. On your own, modify the two location dimensions to **0.0** as shown in the figure below.



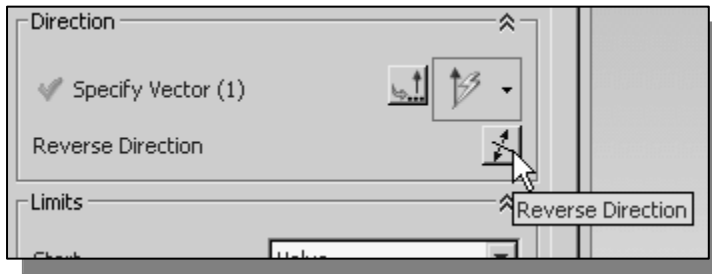
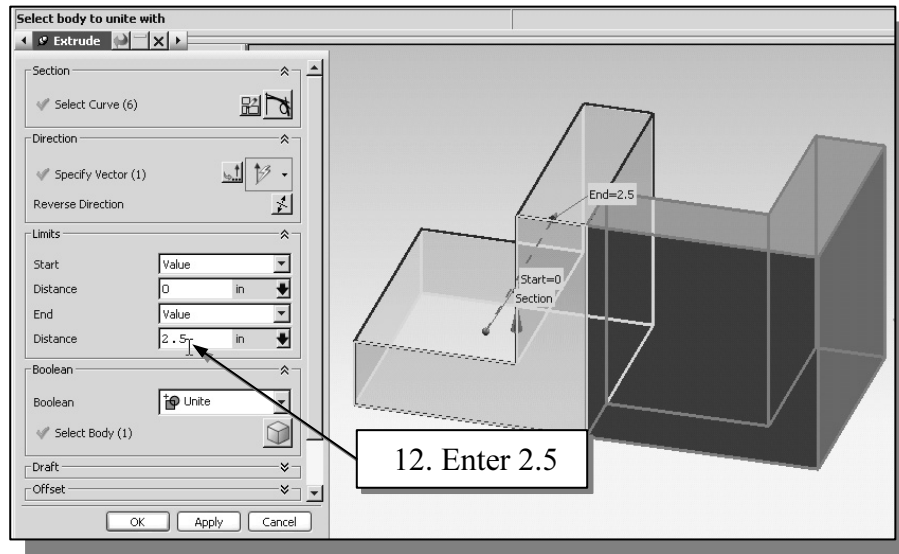
10. Select **Finish Sketch** by clicking once with the left-mouse-button (MB1) on the icon.



11. In the *Extrude* popup window, select **Unite** as the *Extrude Boolean* option.

- The **Unite** option will **add** the new feature to the existing solid. Note that the default option was set to **Create**, which would create a new solid model.

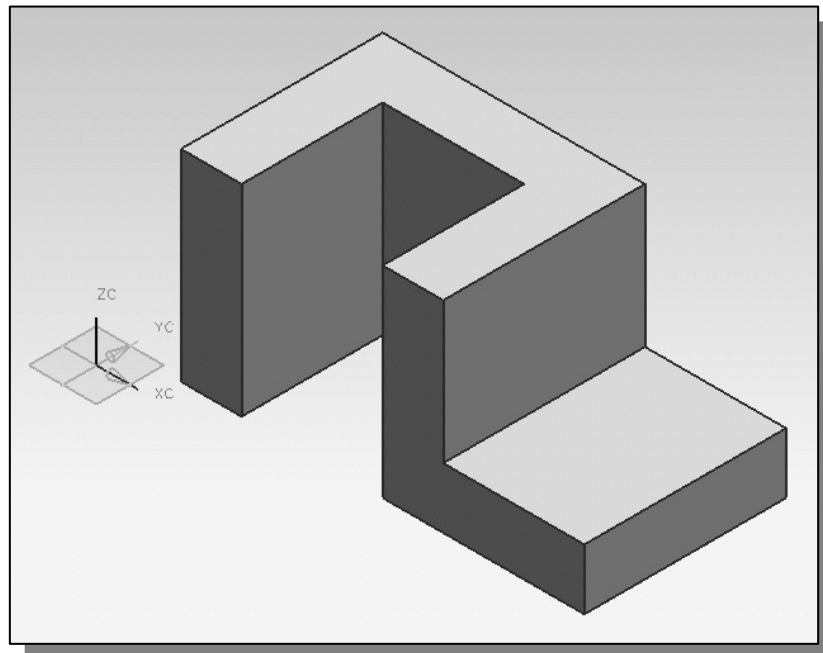
12. In the *Extrude* popup window, enter **2.5** as the extrusion *Distance*.



13. Click on the **Reverse Direction** icon to toggle the extrusion direction so that the extrusion appears as shown in the figure above.



14. Click on the **OK** button to proceed with creating the 3D feature.

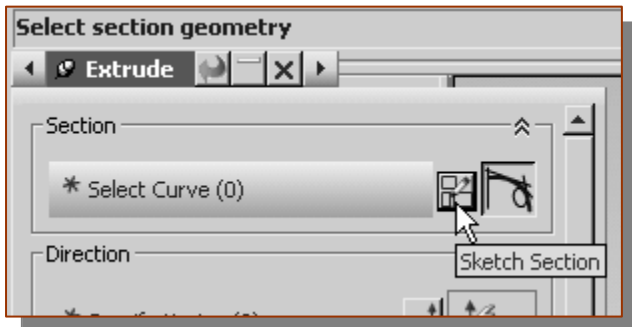


Step 6-2: Adding a Subtract Feature

- Next, we will create and profile a circle, which will be used to create a **subtract** feature that will be added to the existing solid object.

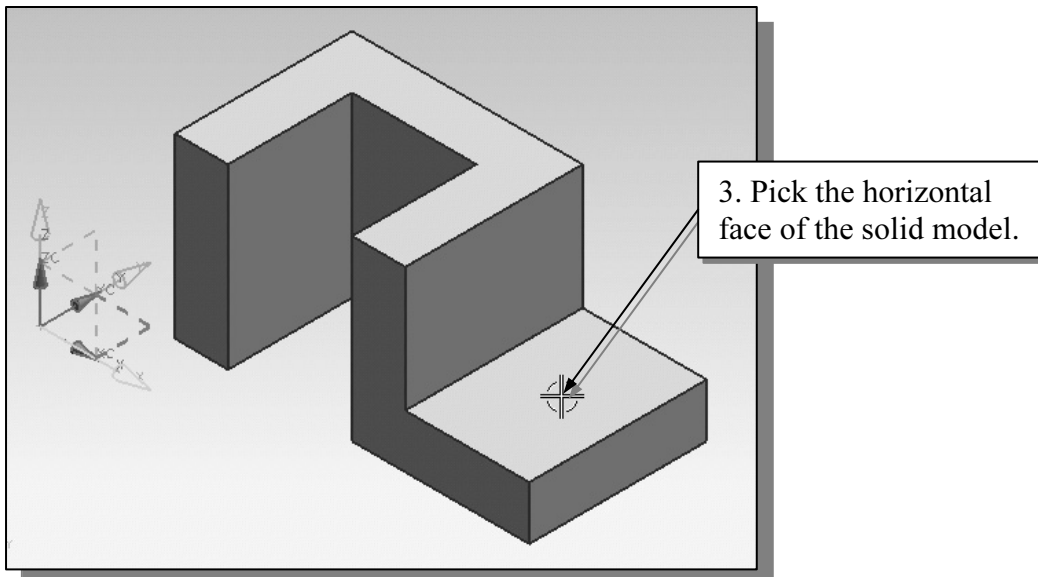


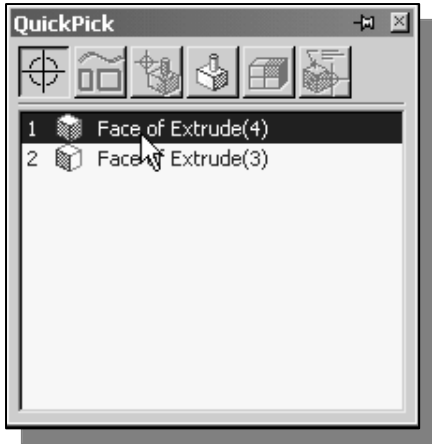
- In the *Feature* toolbars (toolbars aligned to the right edge of the main window), select the **Extrude** icon as shown.



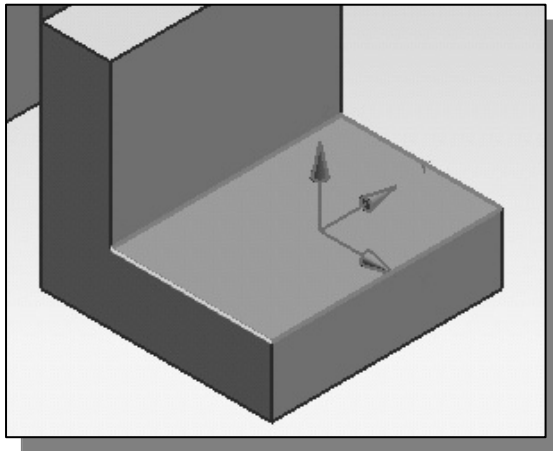
- Click the **Sketch Section** button to begin creating a new *2D sketch*.

- In the *Message* area, the message: “*Select planar face to sketch or select section geometry*” is displayed. *UGS NX* expects us to identify a planar surface where the 2D sketch of the next feature is to be created. Move the graphics cursor on the 3D part and notice that *UGS NX* will automatically highlight feasible planes and surfaces as the cursor is on top of the different surfaces. Pick the horizontal face of the 3D solid object as shown.

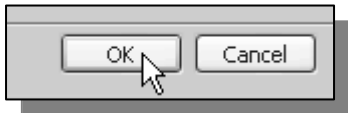




4. Note that if more than one option is feasible for the selection, a **QuickPick** dialog box will appear on the screen. Move the cursor over the different items in the displayed list to examine the feasible items. Click on the desired item to select it.



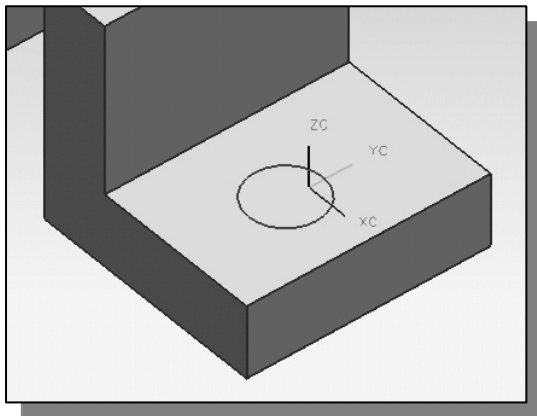
- Note that the *sketch plane* is aligned to the selected face. *UGS NX* automatically establishes a new Work Coordinate System (WCS), and records its location with respect to the part on which it was created.



5. Click **OK** to accept the selected face and proceed with the new WCS.

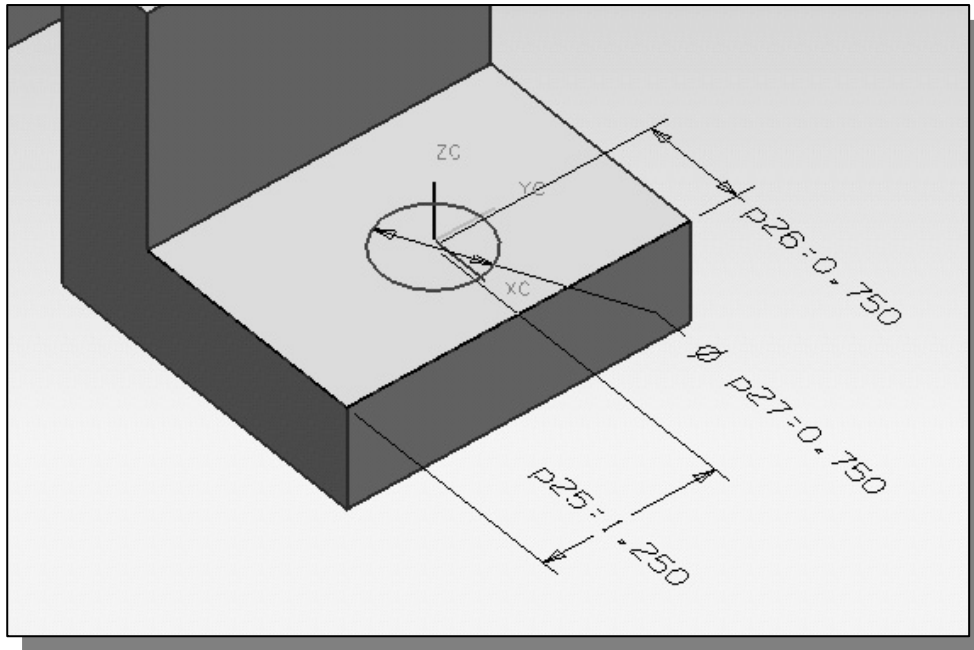


6. Select the **Circle** command by clicking once with the **left-mouse-button (MB1)** on the icon in the *Sketch Curve* toolbar.

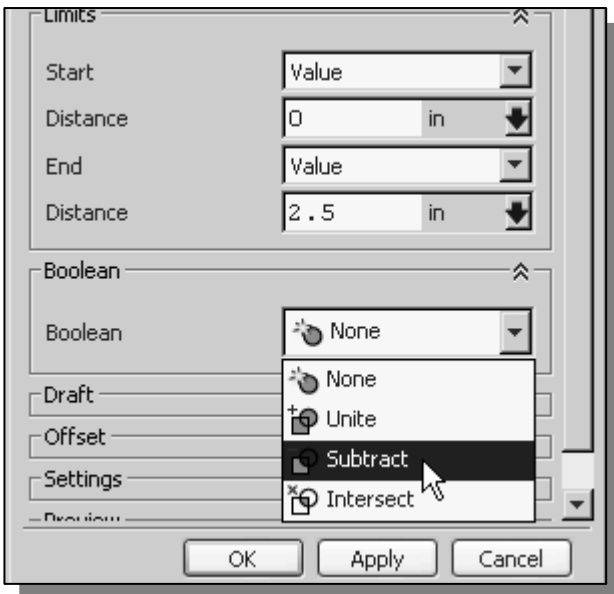


7. Create a circle of arbitrary size on the horizontal face of the solid model as shown.

- On your own, create and modify the dimensions of the sketch as shown in the figure. (Hint: Use two locational dimensions and one size dimension.)

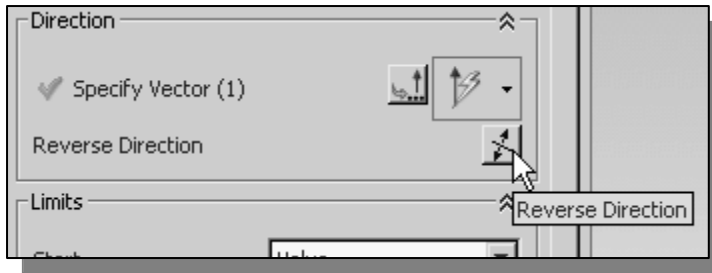


- Select **Finish Sketch** by clicking once with the left-mouse-button (MB1) on the icon.

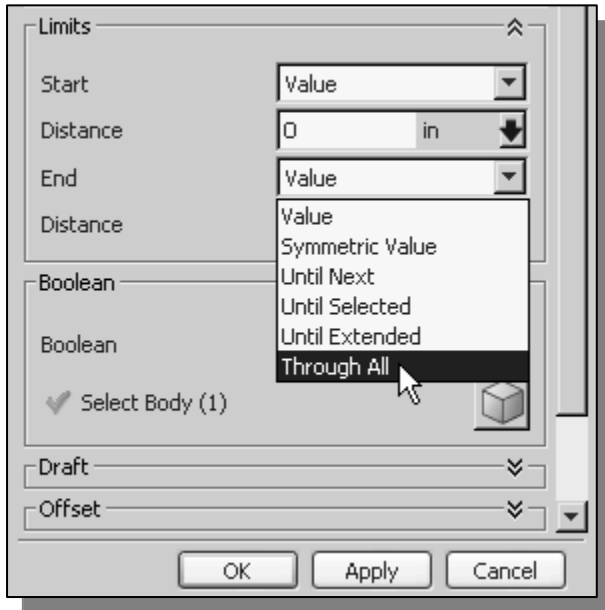


- In the *Extrude* popup window, select **Subtract** as the *Extrude Boolean* option.

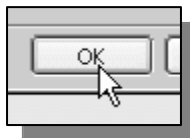
➤ The **Subtract** option will **remove** the volume of the new feature from the existing solid. Note that the default option was set to **Create**, which would create a new solid model.



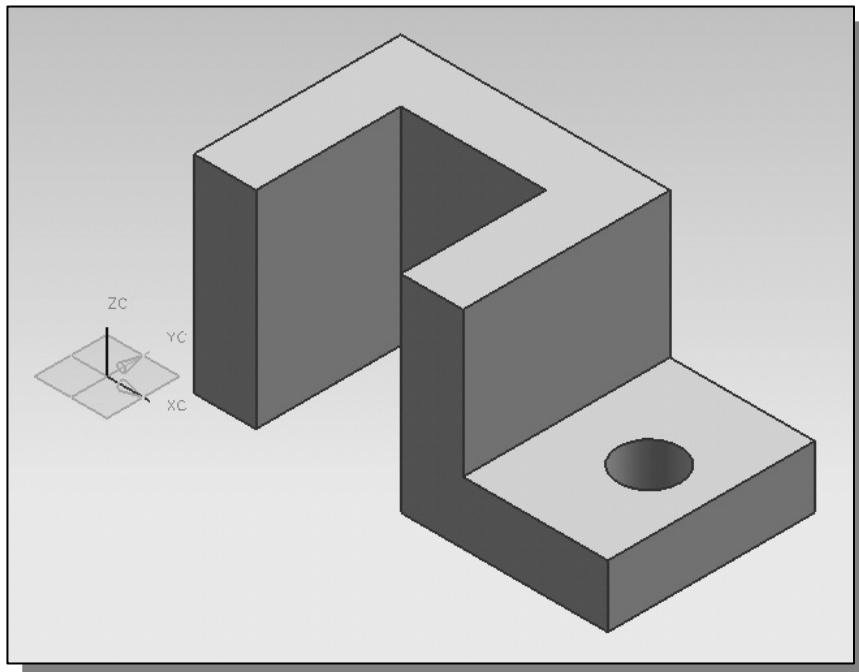
11. Click on the **Reverse Direction** icon to toggle the extrusion direction so that the extrusion appears as shown in the figure.



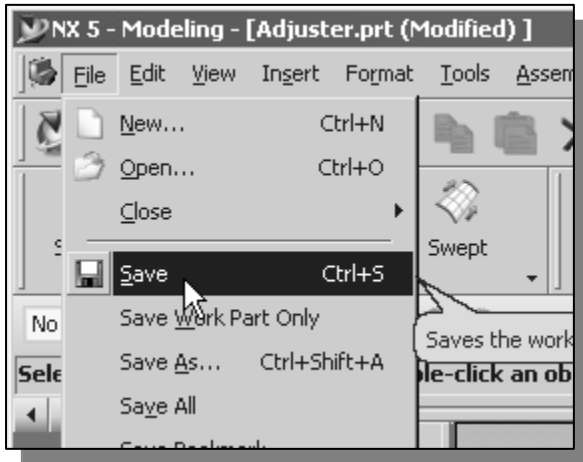
12. In the *Extrude* popup window, choose the **Through All** option to set the *End* extrusion limit.



13. Click on the **OK** button to proceed with creating the 3D feature.

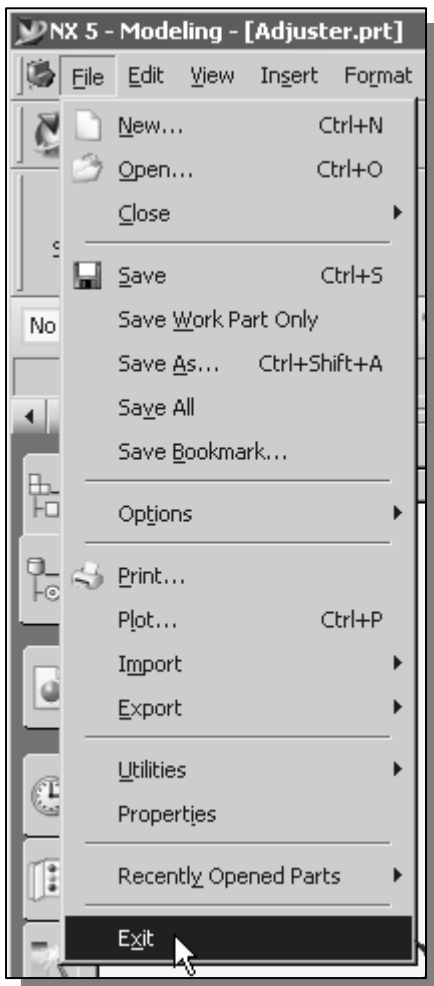


Save the Model and Exit *UGS NX*



1. Select **Save** in the *File* pull-down menu, or you can also use the “**Ctrl-S**” combination (hold down the “**Ctrl**” key and hit the “**S**” key once) to save the part.

- ❖ You should form a habit of saving your work periodically, just in case something might go wrong while you are working on it. In general, one should save one’s work at an interval of every 15 to 20 minutes. One should also save before making any major modifications to the model.



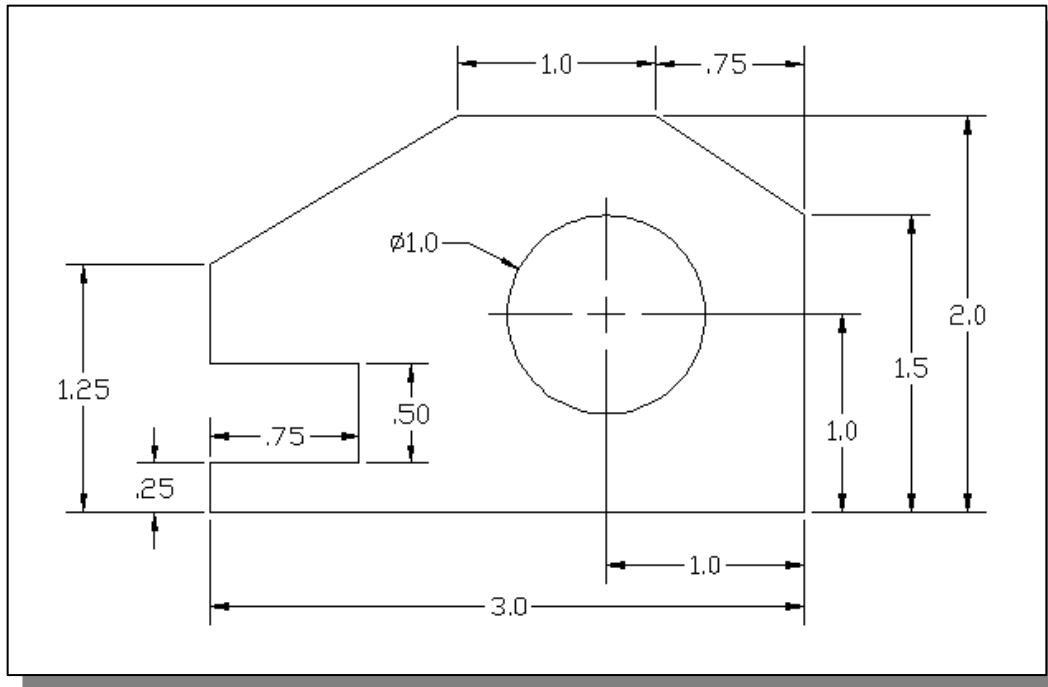
2. Select **Exit** in the *File* pull-down menu to exit *UGS NX*.

Questions:

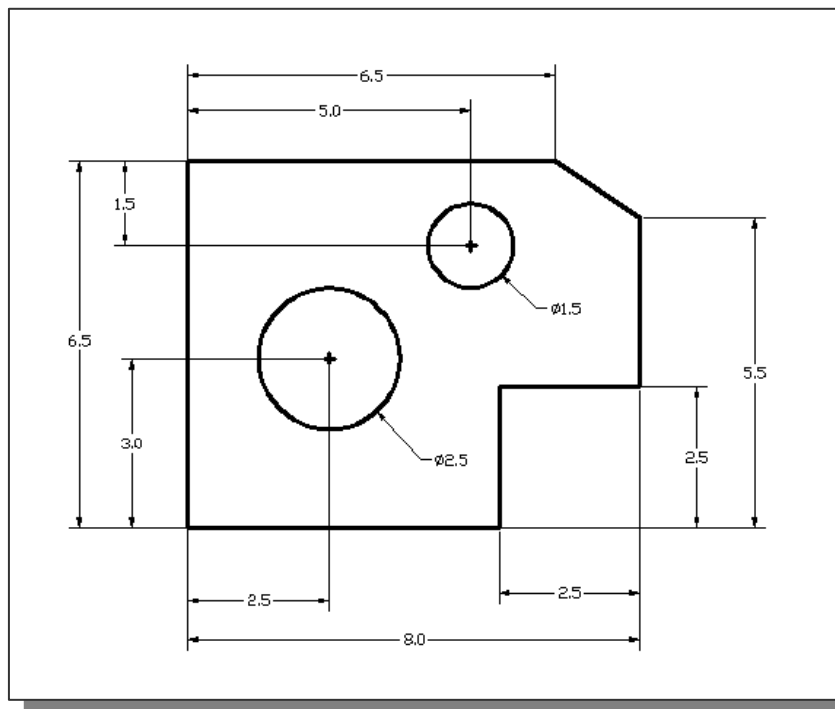
1. What is the first thing we should set up in *UGS NX* when creating a new model?
2. Describe the general *parametric modeling* procedure.
3. List two of the geometric constraint symbols used by *UGS NX*.
4. What was the first feature we created in this lesson?
5. Describe the steps required to define the orientation of the *sketching plane*?
6. How do we change the size of 2D geometry in the *2d Sketcher* mode?

Exercises: (All dimensions are in inches.)

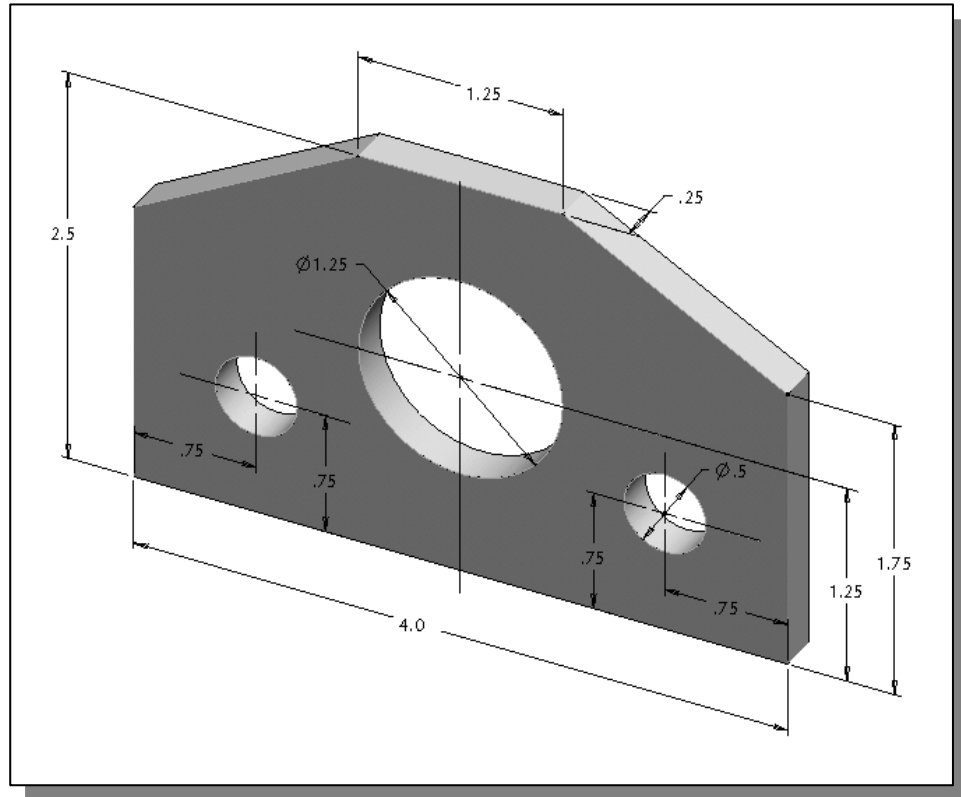
1. Plate Thickness: **.25**



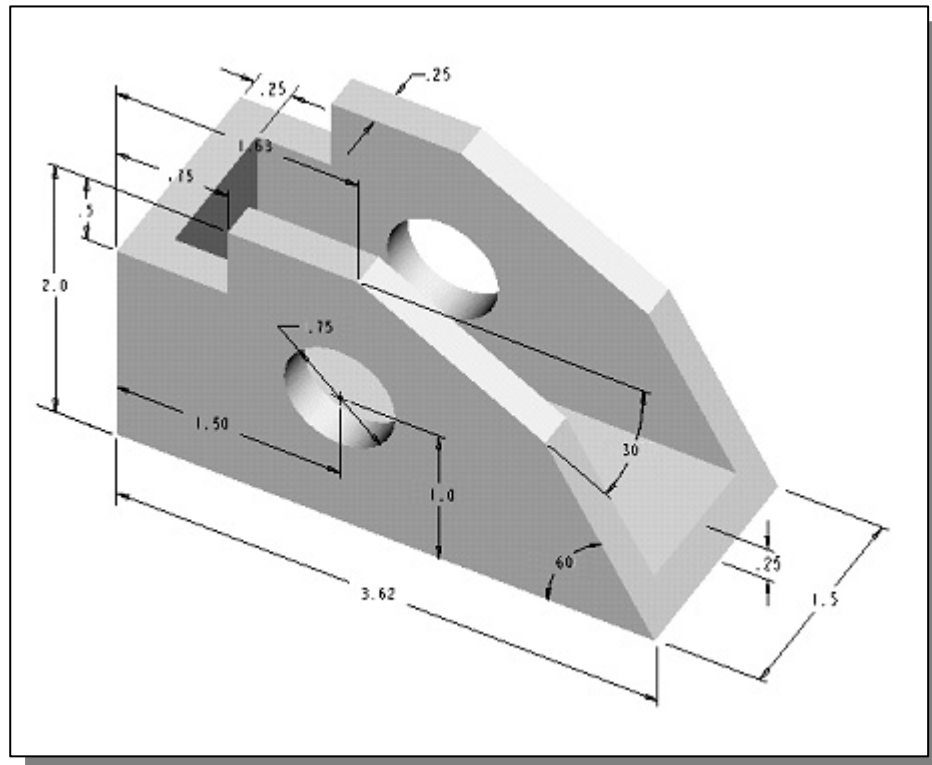
2. Plate Thickness: **.5**



3.



4.



Notes: