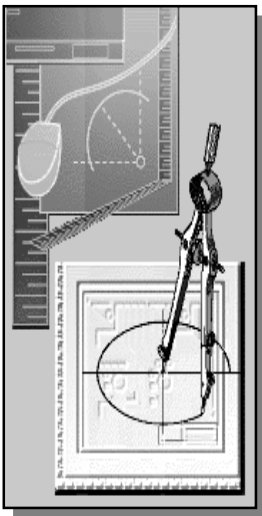
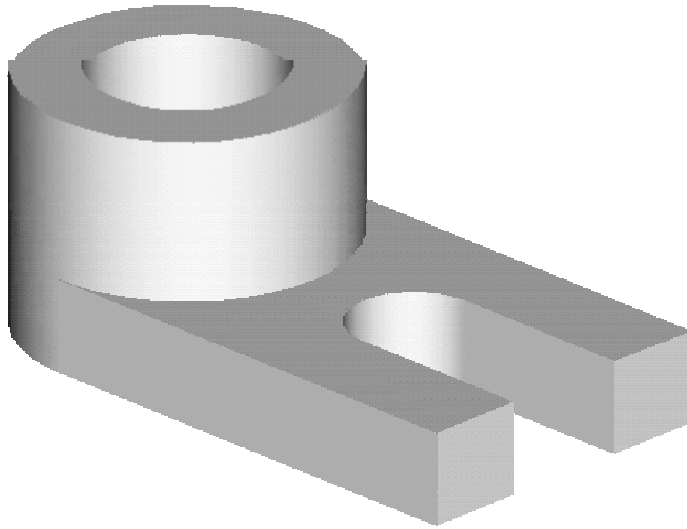


Lesson 2

Constructive Solid Geometry Concept



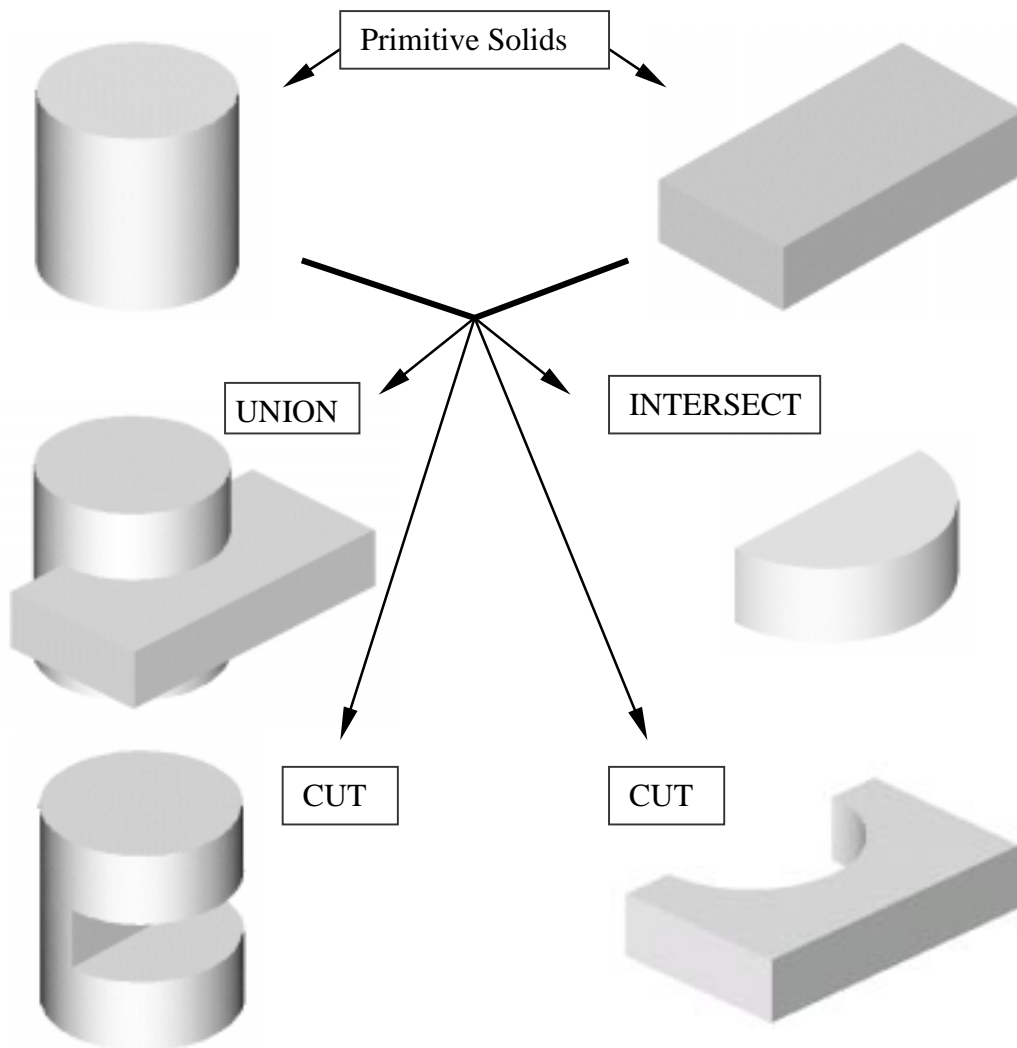
Learning Objectives

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

- ◆ Understand the Basic CSG concepts.
- ◆ Create CSG Binary Trees.
- ◆ Perform the Basic Boolean Operations.
- ◆ Use the Move command.
- ◆ Create Basic Solid Primitives.
- ◆ Use the Catalog Parts.
- ◆ Create Models using Metric Units.

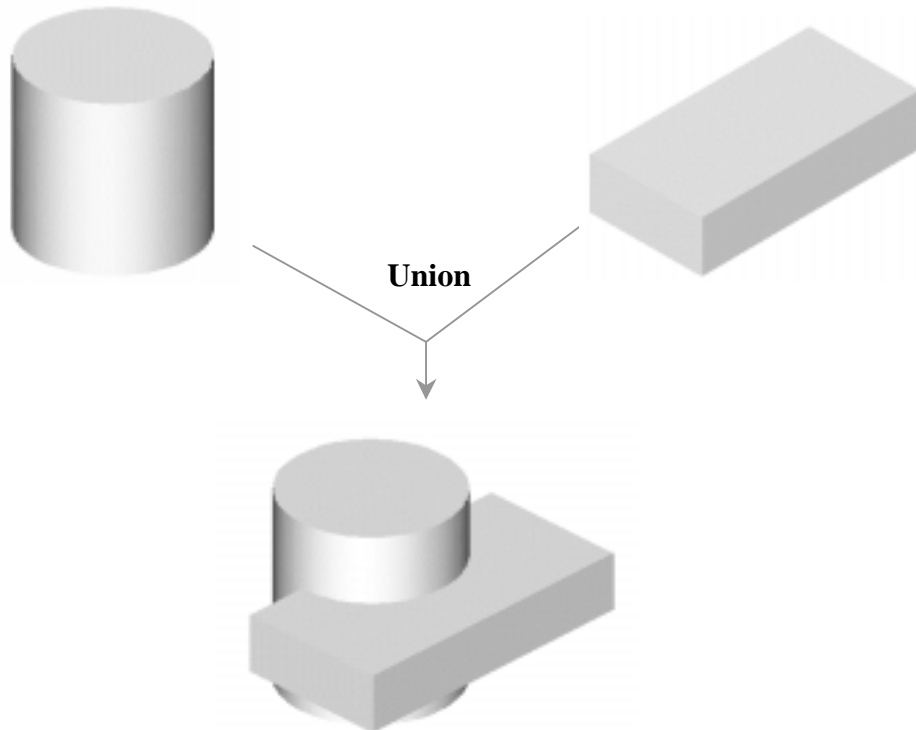
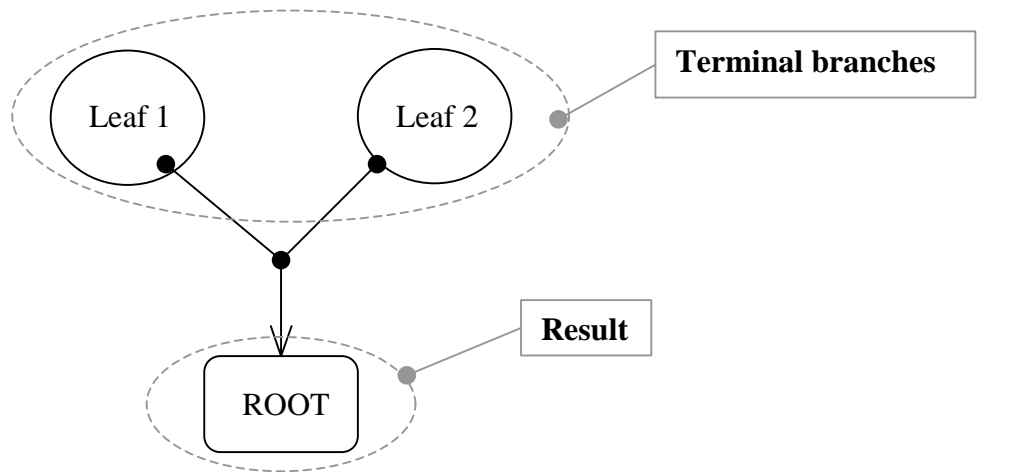
Introduction

In the 1980s, one of the main advancements in **Solid Modeling** was the development of the **Constructive Solid Geometry (CSG)** method. CSG describes the solid model as combinations of basic three-dimensional shapes (**Primitive Solids**). The basic primitive solid set typically includes: Rectangular-prism (Block), Cylinder, Cone, Sphere, and Torus (Tube). Two solid objects can be combined into one object in various ways, and these operations are known as **Boolean operations**. There are three basic Boolean operations: **JOIN (Union)**, **CUT (Difference)**, and **INTERSECT**. The JOIN operation combines the two volumes included in the different solids into a single solid. The CUT operation subtracts the volume of one solid object from the other solid object. The INTERSECT operation keeps only the volume common to both solid objects. The CSG method is also known as the **Machinist's Approach**, as the method is parallel to machine shop practices.



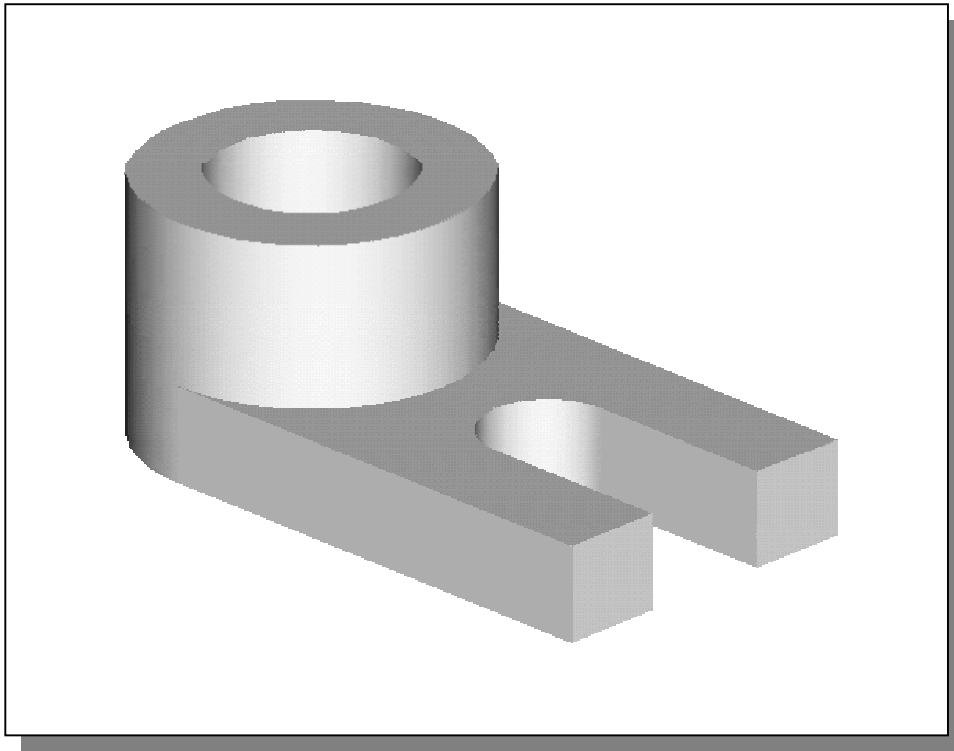
Binary Tree

The CSG is also referred to as the method used to store a solid model in the database. The resulting solid can be easily represented as what is called a **binary tree**. In a binary tree, the terminal branches (leaves) are the various primitives that are linked together to make the final solid object (the root). The binary tree is an effective way to keep track of the *history* of the resulting solid. By keeping track of the history, the solid model can be rebuilt by re-linking through the binary tree. This provides a convenient and intuitive way of modeling that imitates the manufacturing process. We can make modifications at the appropriate links in the binary tree and re-link the rest of the history tree without building a new model.



The Locator Design

The CSG concept is one of the important building blocks for feature-based modeling. In *I-DEAS*, the CSG concept can be used as a planning tool to determine the number of features that are needed to construct the model. It is also a good practice to create features that are parallel to the manufacturing process required for the design. With parametric modeling, we are no longer limited to using only the predefined basic solid shapes. In fact, any solid features we create in *I-DEAS* are used as primitive solids; parametric modeling allows us to maintain full control of the design variables that are used to describe the features. In this lesson, a more in-depth look at the parametric modeling procedure is presented. The equivalent CSG operation for each feature is also illustrated.



- Before going through the tutorial, on your own, make a sketch of a CSG binary tree of the *Locator* design using only two basic types of primitive solids: cylinder and rectangular prism. In your sketch, how many *Boolean operations* will be required to create the model? What is your choice of the first primitive solid to use, and why? Take a few minutes to consider these questions and do the preliminary planning by sketching on a piece of paper. Compare the sketch you make to the CSG binary tree steps shown on page 2-6. Note that there are many different possibilities in combining the basic primitive solids to form the solid model. Even for the simplest design, it is possible to take several different approaches to creating the same solid model.

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. Start a new model file by filling in the window items as shown below in the *I-DEAS Start* window:

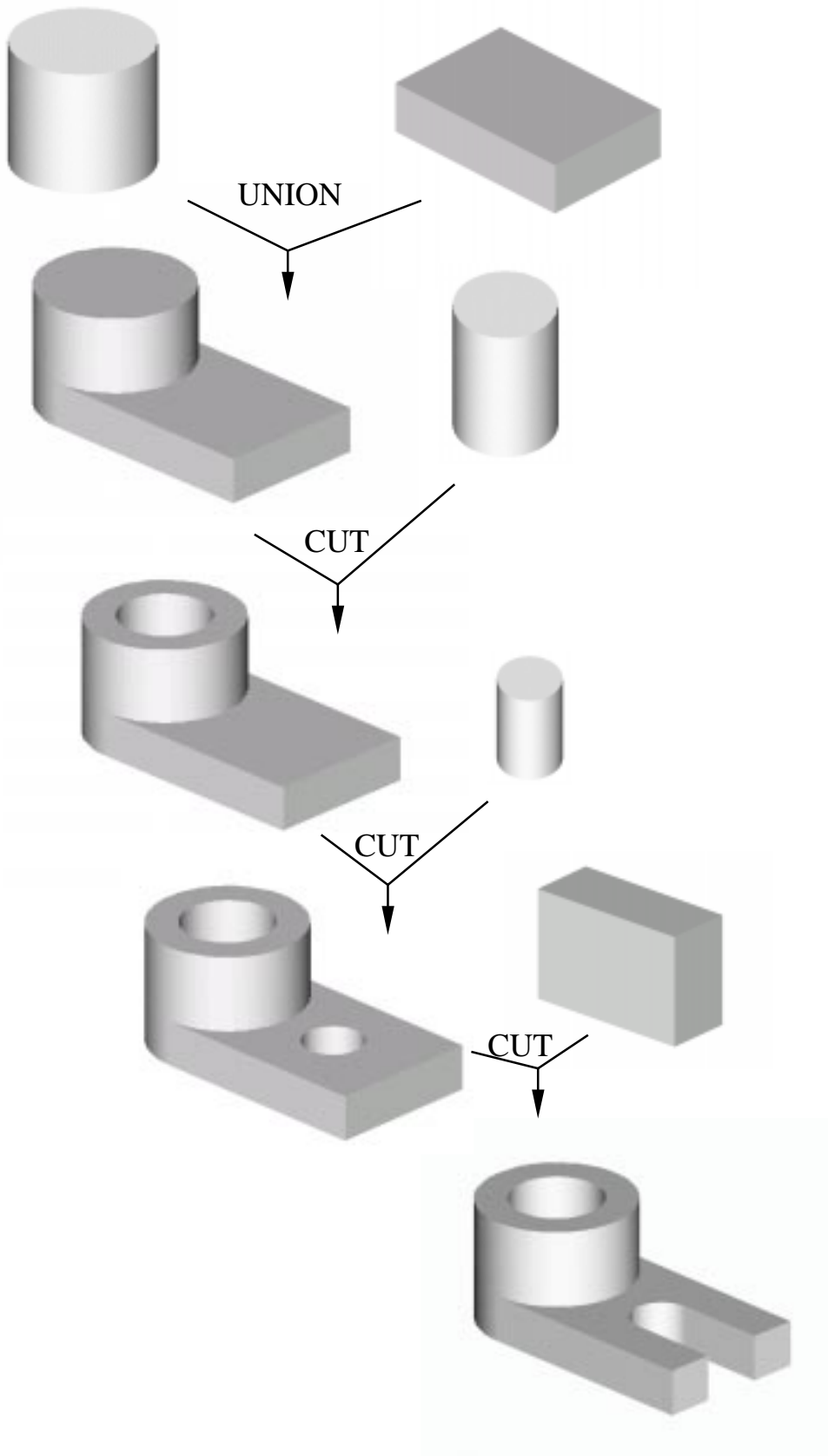
Project Name: (Your account Name) Model File Name: Locator Application: Design Task: Master Modeler <input type="button" value="OK"/>

3. After you click **OK**, a *warning window* will appear to tell you that a new model file will be created. Click **OK** to continue.

I-DEAS Warning
! New Model File will be created
<input type="button" value="OK"/>

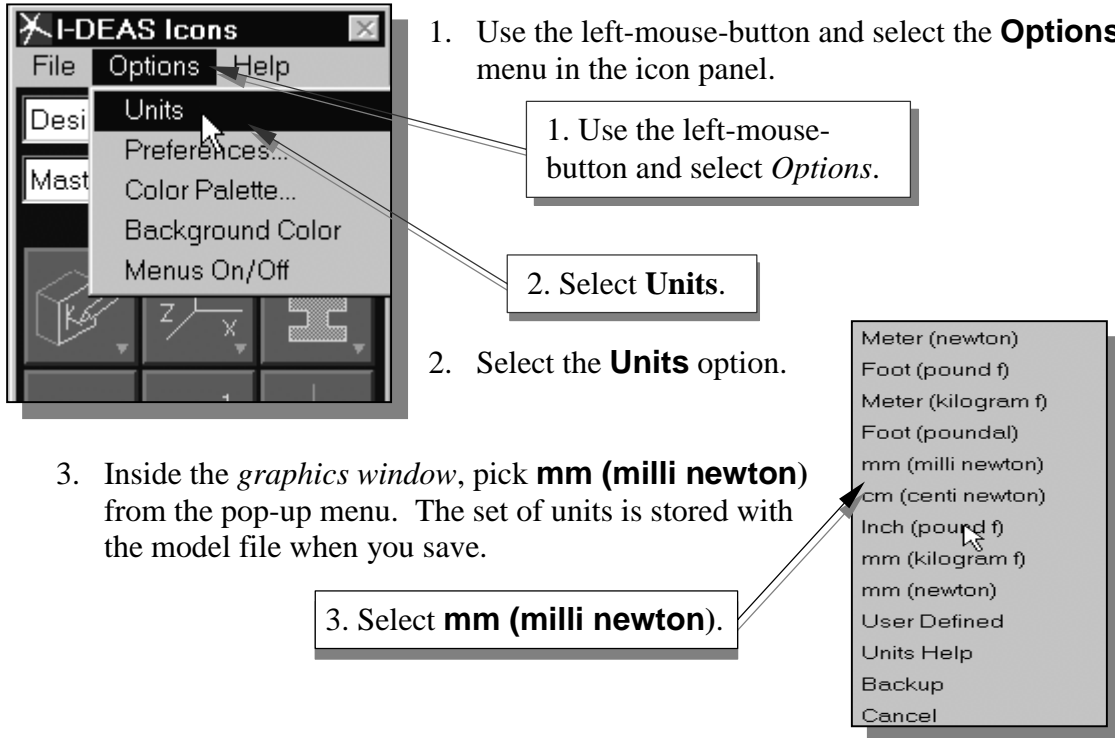
Next, I-DEAS will display four windows, the *graphics window*, the *prompt window*, the *list window*, and the *icon panel*. A line of *quick help* text appears at the bottom of the *graphics window* as you move the mouse cursor over the icons.

The CSG Binary Tree of the Locator Design



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.

1. Use the left-mouse-button and select *Options*.

2. Select **Units**.

2. Select the **Units** option.

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

3. Select **mm (milli newton)**.

Meter (newton)
Foot (pound f)
Meter (kilogram f)
Foot (poundal)
mm (milli newton)
cm (centi newton)
Inch (pound f)
mm (kilogram f)
mm (newton)
User Defined
Units Help
Backup
Cancel

Base Feature

In *parametric modeling*, the first solid feature is called the **base feature**, which usually is the primary shape of the model. Depending upon the design intent, additional features are added to the base feature.

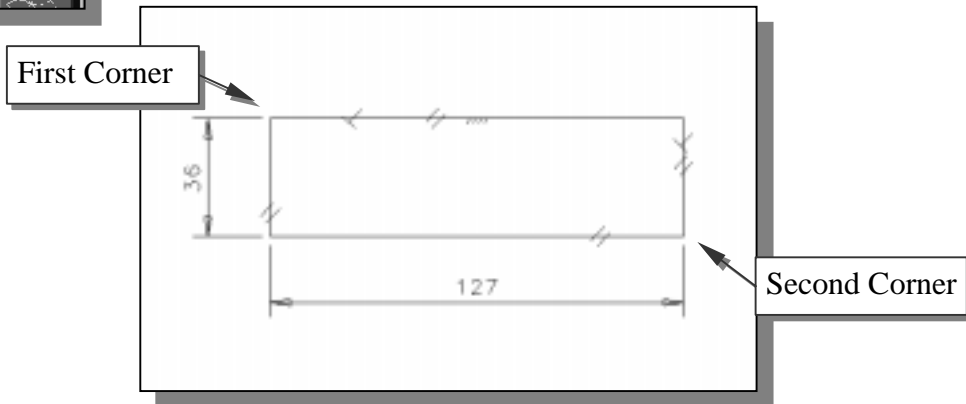
Some of the considerations involved in selecting the base feature:

- **Design Intent** – Determine the functionality of the design; identify the feature that is central to the design.
- **Order of features** – Choose the feature that is the logical base in terms of the order of features in the design.
- **Ease of making modifications** – Select the base feature that is more stable and is less likely to be changed.

We will create a rectangular block as the base feature of the *Locator* design.



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 of arbitrary size by selecting two locations on the screen as shown below.



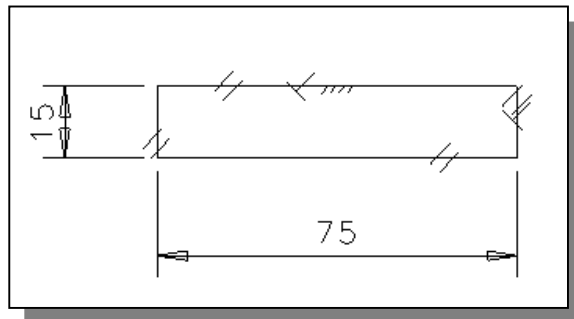
Modifying the Size of the Rectangle

1. *Pre-select* the two dimensions of the rectangle by holding down the SHIFT key and left clicking the values. The selected dimension will be highlighted.

PRE-SELECT



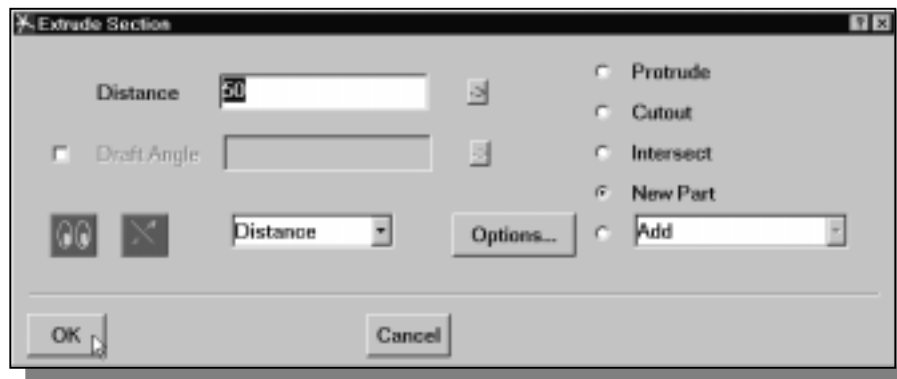
2. Choose **Modify** in the icon panel. The *Dimensions* window appears.
3. In the *Dimensions* window, change the values to the values as shown (15 X 75).



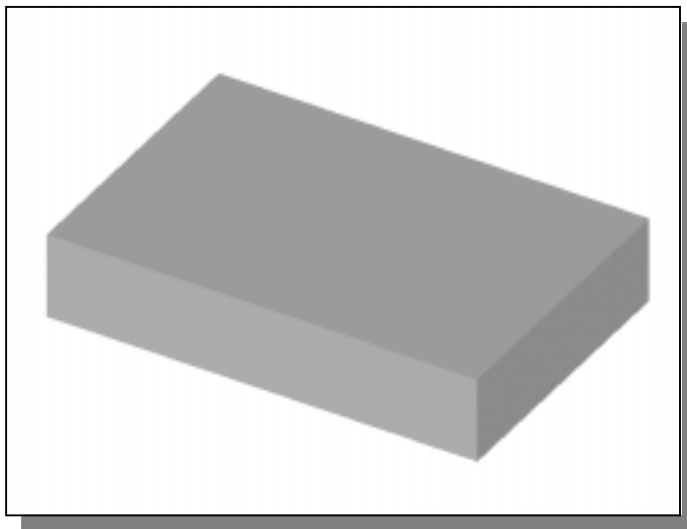
Completing the Base Solid Feature



1. Choose **Extrude** in the icon panel. In the prompt window, the message “*Pick curve or section*” is displayed.
2. Pick the rectangle at any location.
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 will appear on the screen. Enter **50** as the *extrusion distance* and confirm that the *New part* option is switched **on** as shown.



5. Click on the **OK** button to accept the settings and extrude the 2-D 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.

Creating the next solid feature

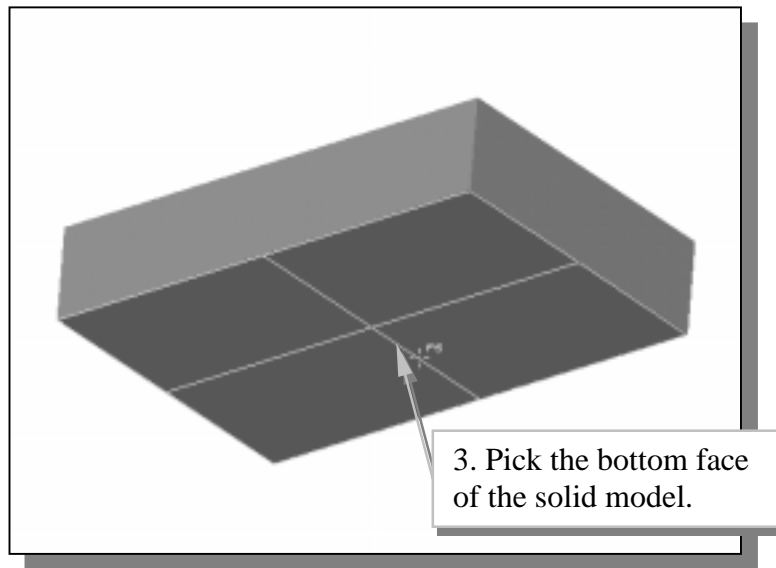


1. Choose **Sketch in Place** in the icon panel. In the prompt window, the message “Pick plane to sketch on” is displayed.

2. Use the dynamic rotation function to display the bottom face of the solid model as shown below.

3. Pick the bottom face of the 3D model when it is highlighted as shown.

➤ Note that the sketch plane is aligned to the selected face. *I-DEAS* automatically establishes a User-Coordinate-System (UCS) and records its location with respect to the part on which it was created.

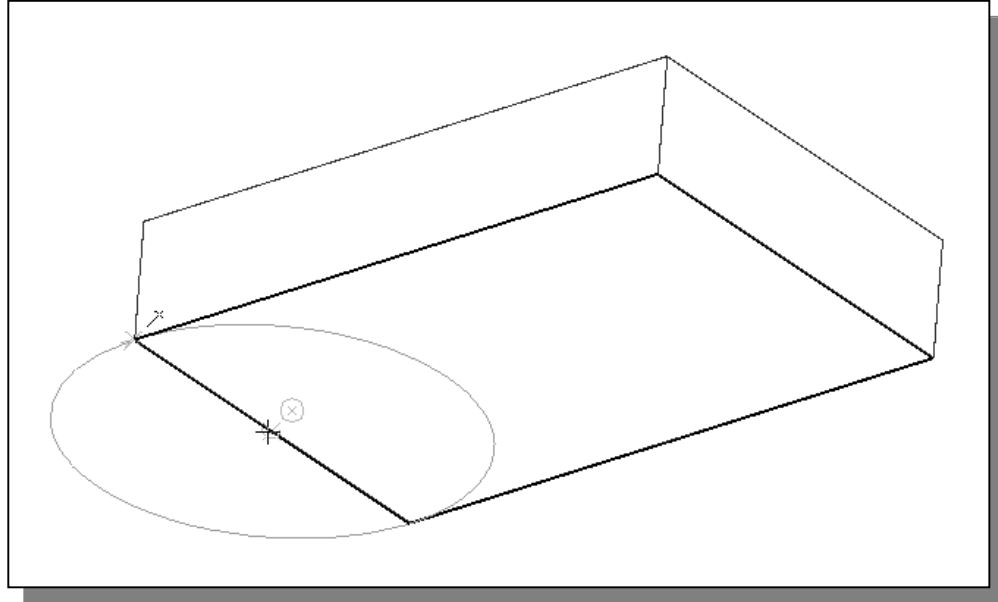


3. Pick the bottom face of the solid model.



4. 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.

5. Move the graphics cursor near the midpoint of the shorter edge of the bottom surface. Notice the *Dynamic Navigator* will automatically show alignment to the center of the edge. Click with the **left-mouse-button** to select the position as the center point of the circle.

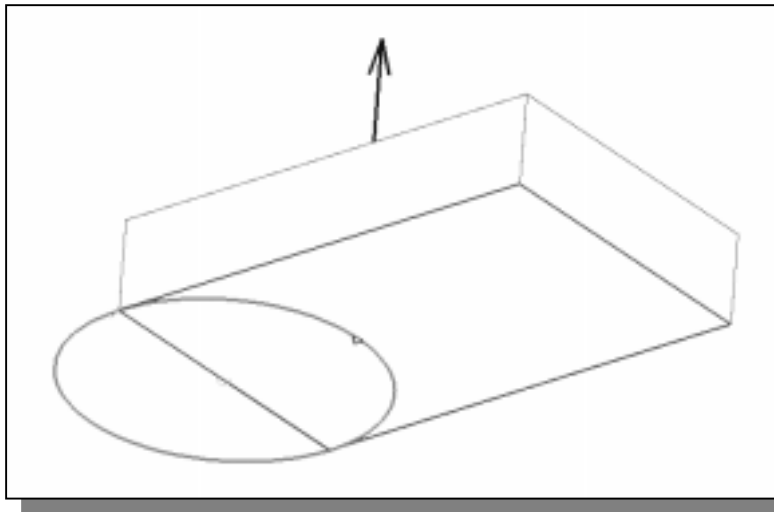
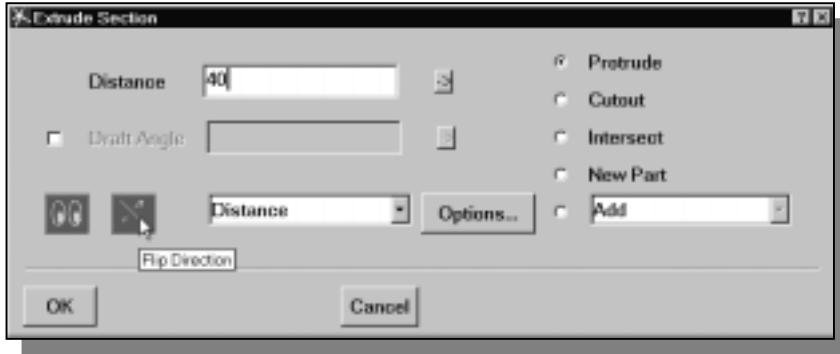


6. Move the graphics cursor near the corner of the bottom surface, click with the **left-mouse-button** when the alignment is displayed as shown. This location is the location where the circle will pass through. Note that the diameter dimension is not necessary since the circle is well defined. If a dimension appears on your screen, delete the circle and create the circle again.
7. Press the **ENTER** key or the **middle-mouse-button** to end the *Circle – Center Edge* command.



8. Choose **Extrude** in the icon panel. In the prompt window, the message "*Pick curve or section*" is displayed.
9. Pick the circle as the 2D section to be extruded.
10. 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.

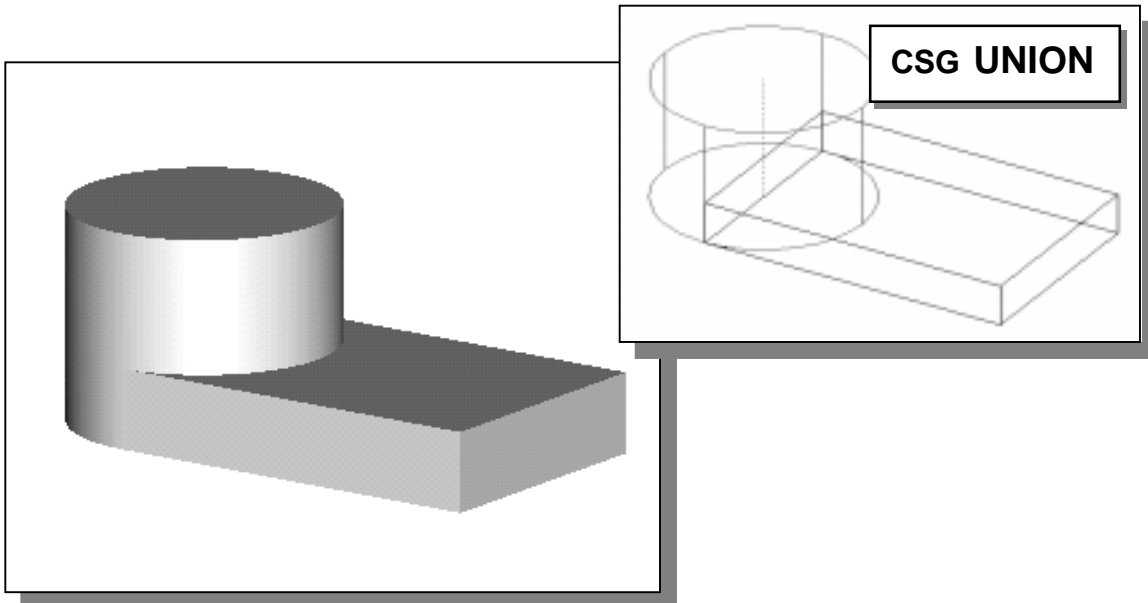
- The *Extrude Section* window will appear on the screen. Enter **40** as the *extrusion distance* and confirm that the *Protrude* option is switched **on** as shown.



- Click on the **Flip Direction** button to reverse the direction of extrusion (upward) as shown below.

- Click on the **OK** button to proceed with the join operation.

- The two features are joined together into one solid part; the **CSG-Union** operation was performed.

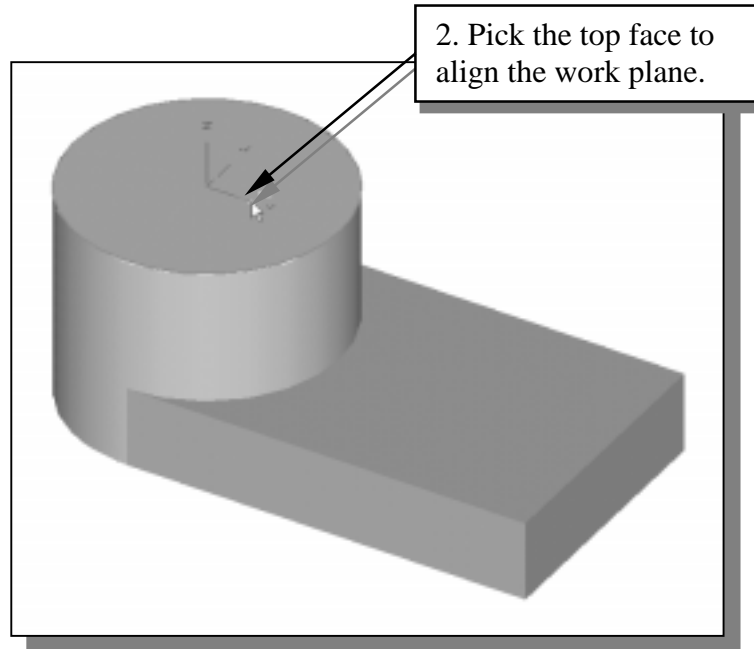


Creating a CUT Feature

- We will create a circular cut as the next solid feature of the design. We will align the sketch plane to the top of the last cylinder feature.



- Choose **Sketch in Place** in the icon panel. In the prompt window, the message "*Pick plane to sketch on*" is displayed.
- Pick the top face of the cylinder as shown.

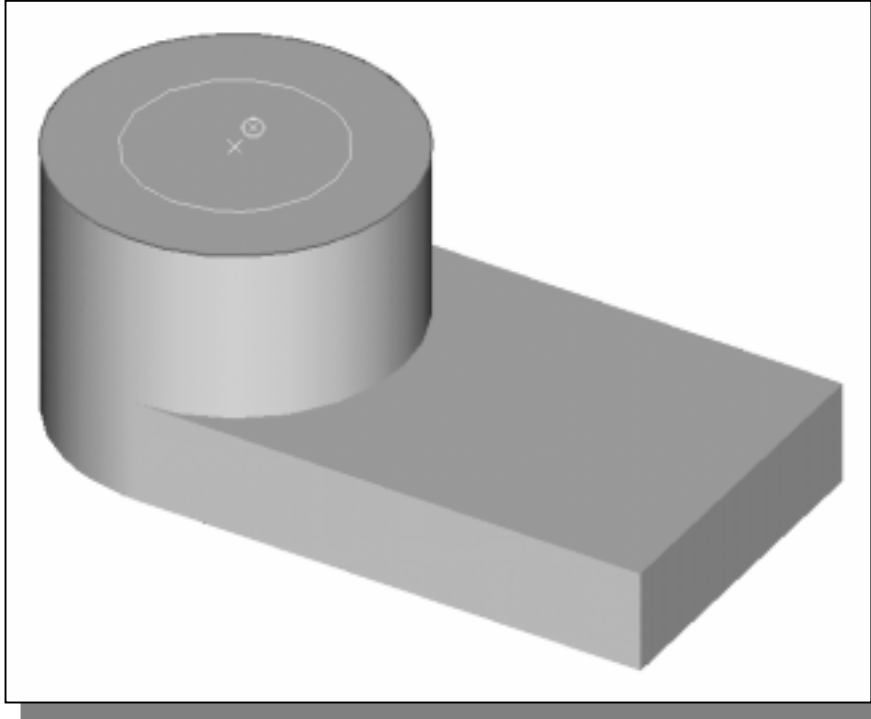


2. Pick the top face to align the work plane.



- 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.

4. Move the graphics cursor near the center of the cylinder of the 3D solid. Notice the *Dynamic Navigator* will automatically show alignment to the center of the surface. Click with the **left-mouse-button** to select the position as the center point of the circle.

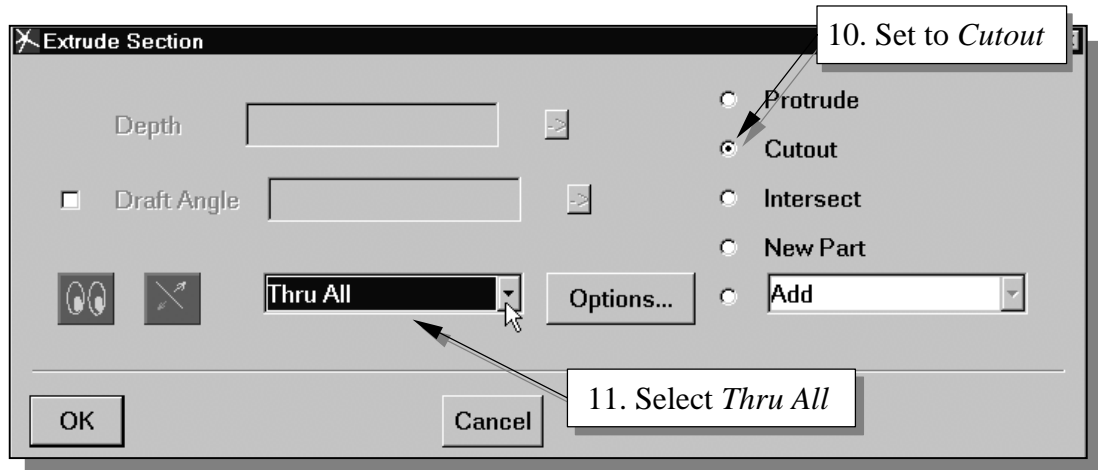


5. Sketch a circle of arbitrary size inside the top face of the cylinder as shown.
6. On your own, modify the dimension to **30** mm.

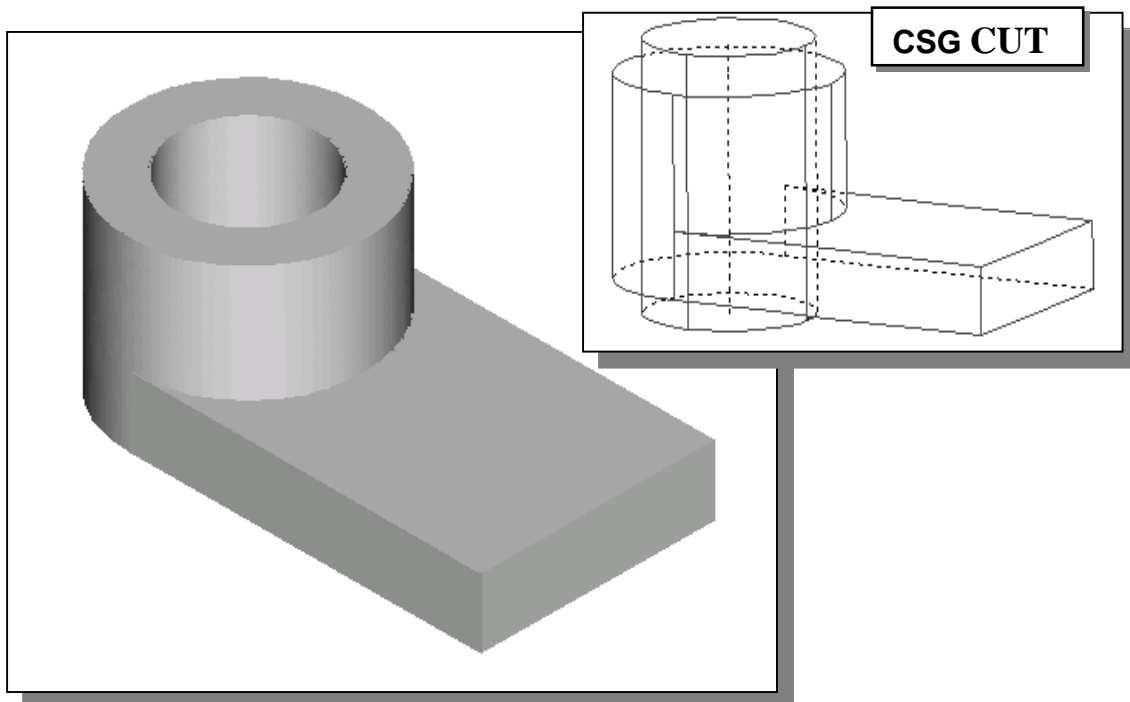


7. Choose **Extrude** in the icon panel. In the prompt window, the message "*Pick curve or section*" is displayed.
8. Pick the circle we just created as the 2D section to be extruded.
9. At the I-DEAS prompt "*Pick curve to add or remove (Done)*," press the **ENTER** key or the **middle-mouse-button** to accept the selection.

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



11. Click on 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.
12. Click on the **OK** button to accept the settings. The circle is extruded into a cylinder and the center of the 3D object is removed.



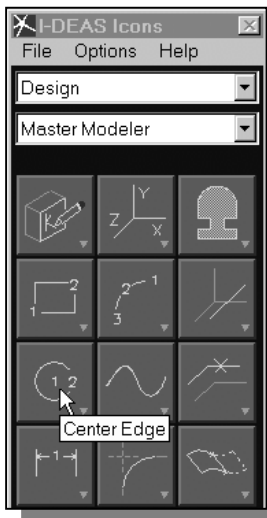
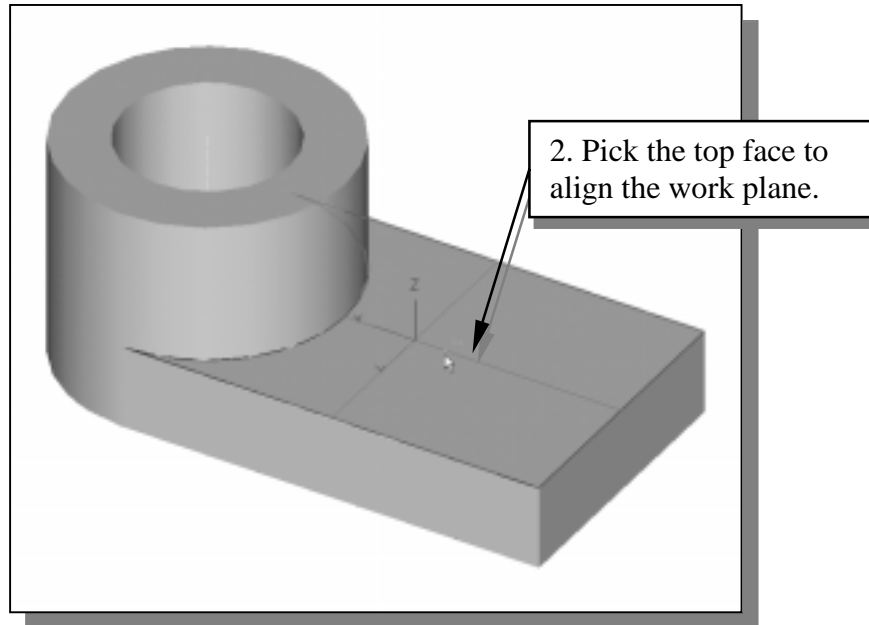
Now is a good time to save the model (Quick key: **[Ctrl] + [S]**). It is a good habit to save your model periodically, just in case something might and probably will go wrong while you are working on the model. You should also save the model after you have completed any major constructions.

The Second Cut Feature

- We will create a circular cut as the next solid feature of the design. We will align the sketch plane to the top of the base feature.

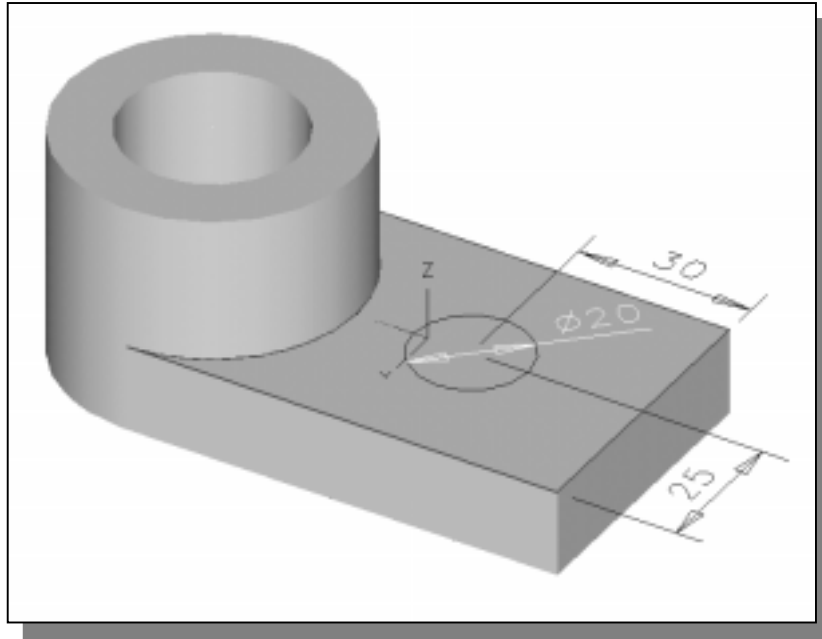


1. Choose **Sketch in Place** in the icon panel. In the prompt window, the message "*Pick plane to sketch on*" is displayed.
2. Pick the top face of the base feature as shown.

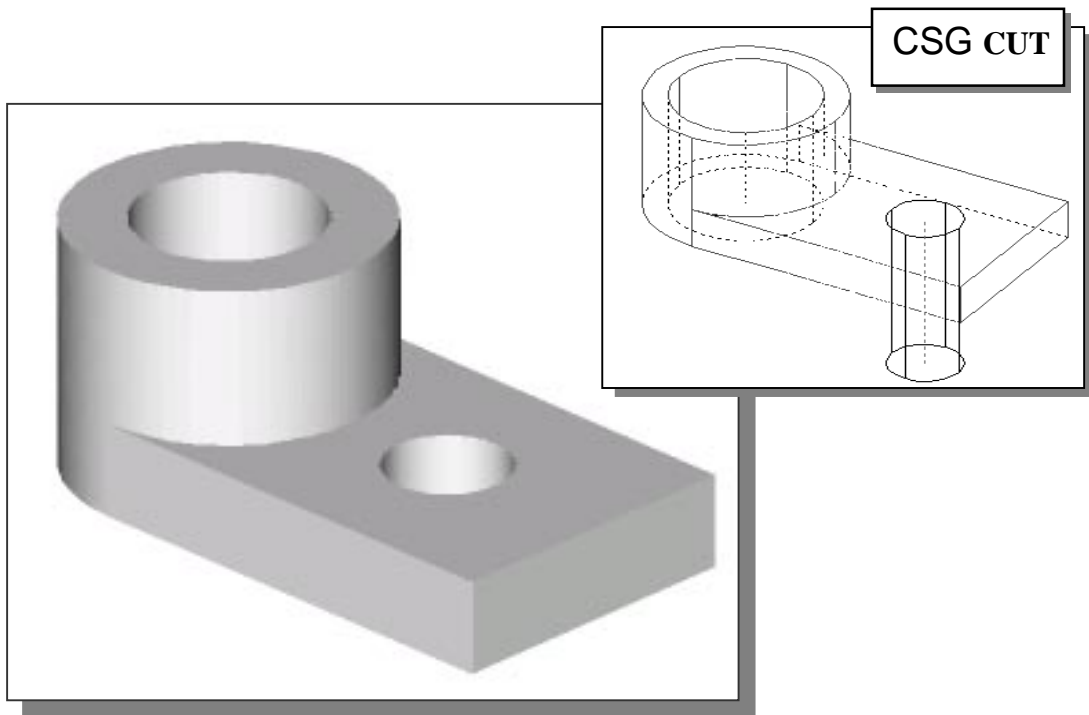


3. Choose **Circle - Center Edge** in the icon panel. This command requires the selection of two locations: first the location of the center of the circle and then a location where the circle will pass through.

4. On your own, create a circle (diameter 20 mm) located at 25 mm and 30 mm measured from the front corner of the base feature as shown below.



5. On your own, use the *Extrude* command and create the cutout through the base of the solid as shown.

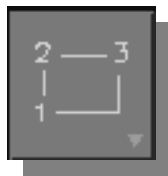
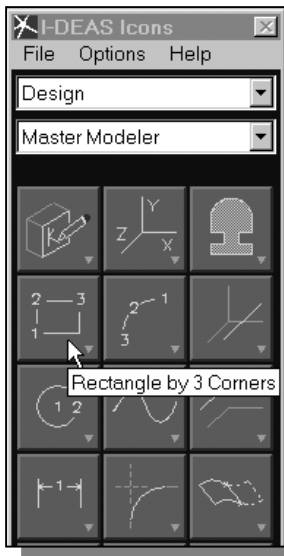
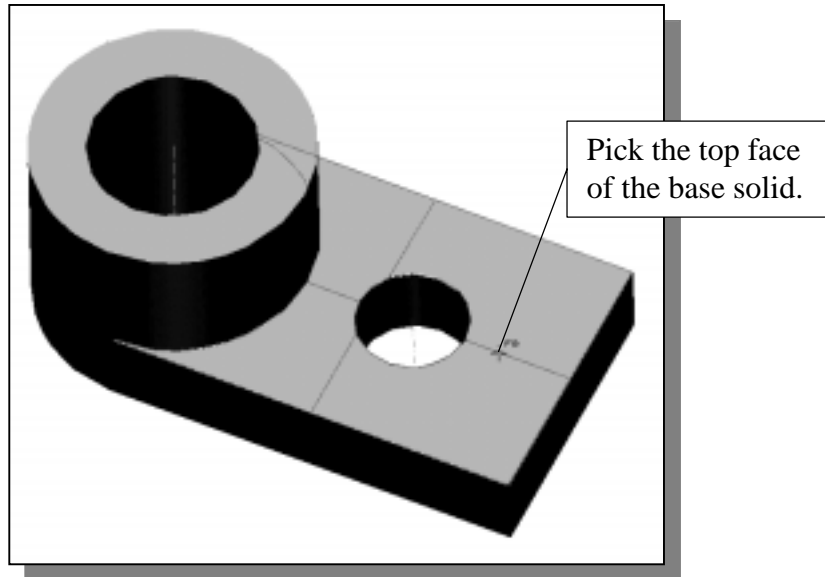


The Third *CUT* Feature

We will create a rectangular cutter as the final solid feature of the part. We will also demonstrate the use of the *Three point Rectangle option*.

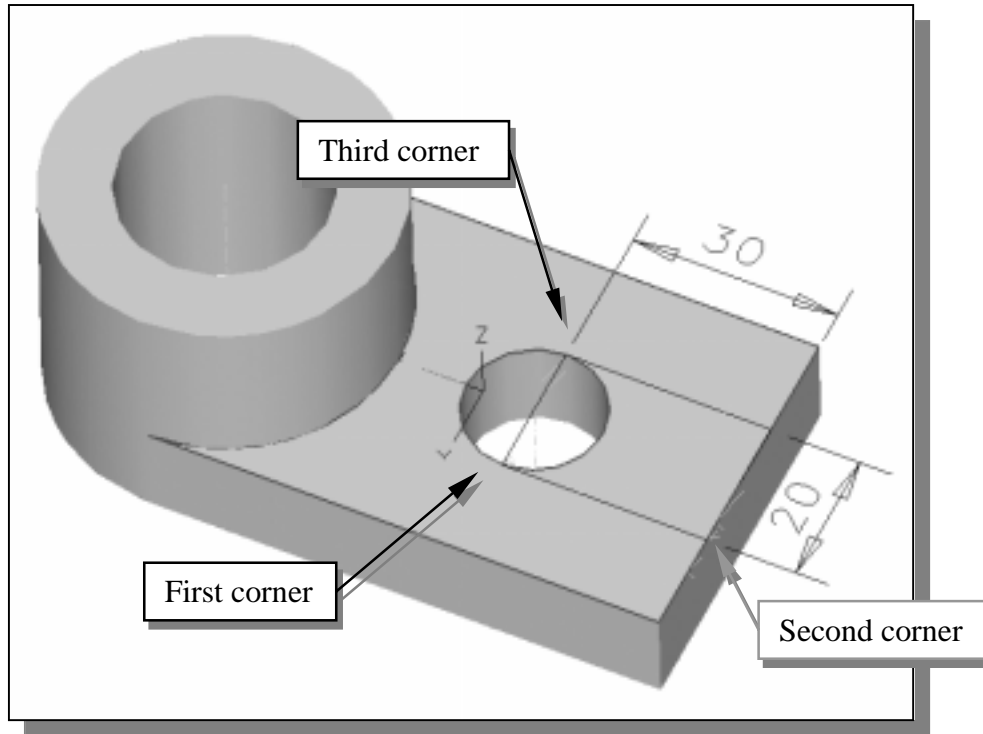


1. Choose **Sketch in Place** in the icon panel. In the prompt window, the message "*Pick plane to sketch on*" is displayed.
2. Pick the top face of the base feature by left-clicking the surface as shown in the below figure.

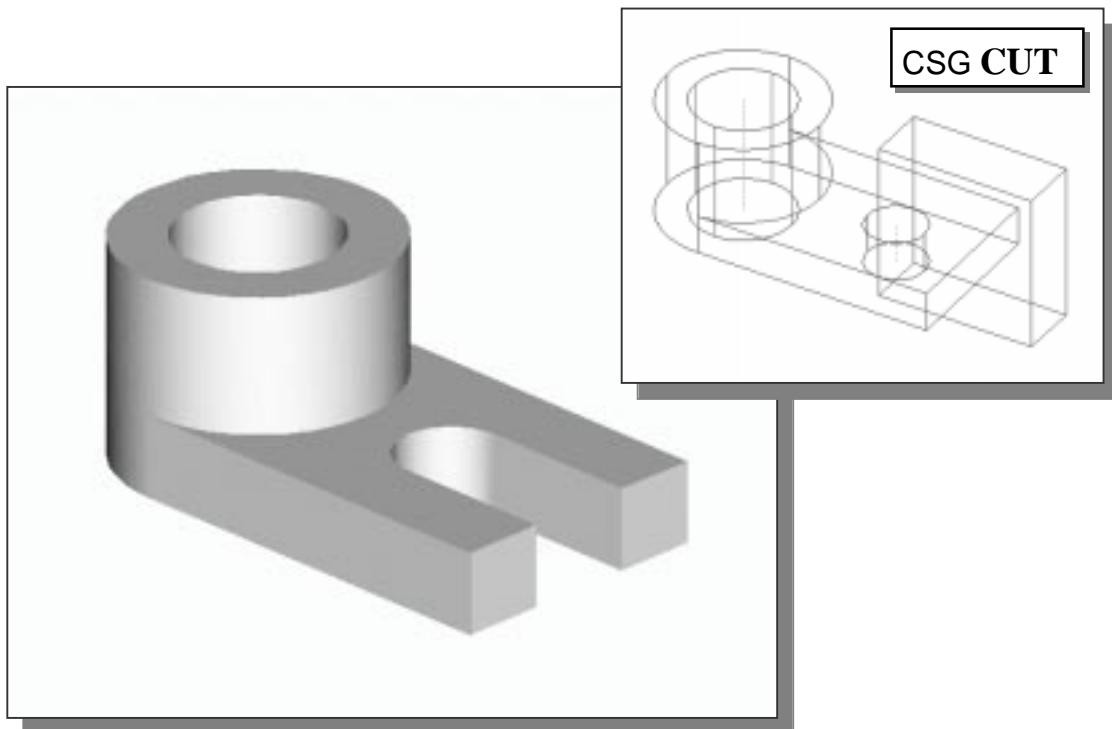


3. Choose **Rectangle by 3 Corners** in the icon panel. This command requires the selection of three locations to identify three specific alignments of a rectangle. The message "*Locate first corner*" is displayed in the prompt window.

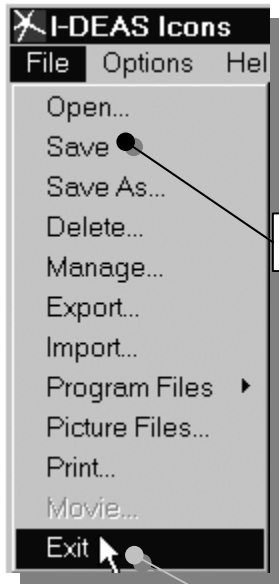
4. Create a rectangle by selecting the three corners as shown. The first and third points are aligned to the circular cut feature.



6. On your own, complete the *cut feature* using the *Thru All* option.



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.

- The CSG concept described in this lesson is the underlying mainstream method that is used in almost all of the available solid modelers today. The solid modeling software performs the **Boolean operations** either implicitly or explicitly. One should always consider the CSG approach in creating solid models; the simplicity of the CSG approach also makes it extremely powerful.

Questions:

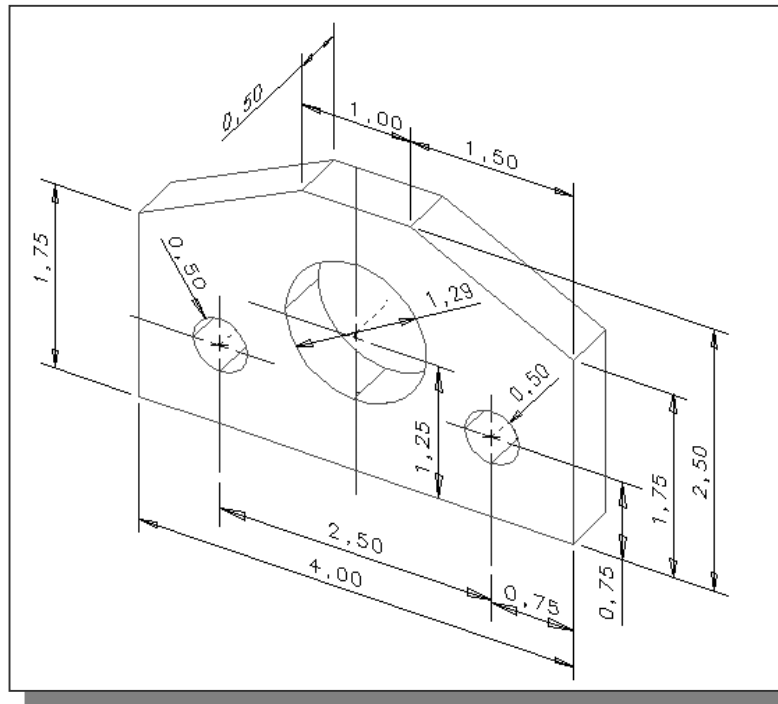
1. List and describe the three basic *Boolean operations* commonly used in computer geometric modeling software?
2. What is a *Primitive Solid*?
3. What does *CSG* stand for?
4. Which Boolean operation keeps only the volume common to the two solid objects?
5. What are the considerations in choosing the base feature of a solid model?
6. Using the CSG concepts, create *Binary Tree* sketches showing the steps you plan to use to create the two models shown on the next page:

Ex. 1)

Ex. 2)

Exercises:

1.



2.

