## 10 Minute Tour of the BobCAD Menus and Features

Before you can begin the steps in Lesson 2, you need to know how to navigate the menus and use the mouse functions. So, let's get started.

Depending on the version of BobCAD you are using, you start the program by clicking on an icon similar to one of the ones shown below.

BobCAD-CAM V19   BobCAD-CAM V20

When you start v19, you are likely to be prompted with the following dialog box.
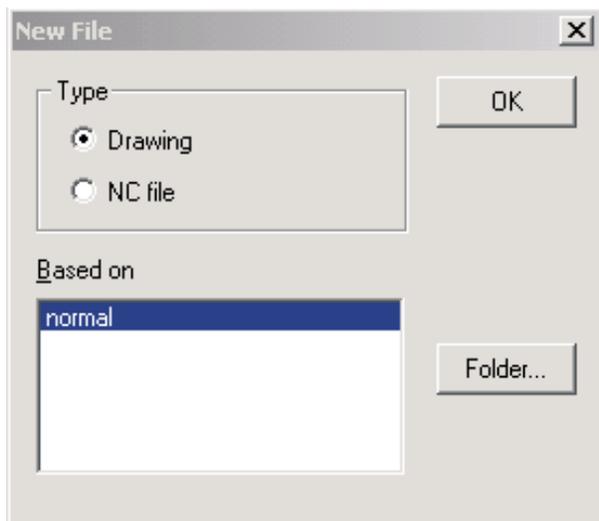
This simply gives you the opportunity to select whether you are going to be working with a CAD file or an NC file. As a general rule, you would always be starting with a CAD file.

In the screen captures shown on the right, you will see the choices available from the top menu when you first start BobCAD.

Shown in the bottom picture is a sample of the help text that is available by hovering the mouse cursor over any icon.
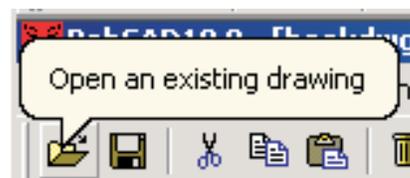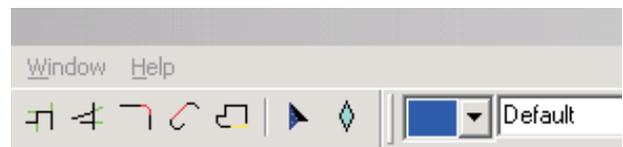
If you are familiar with other CAD/CAM, systems, you will have a good idea of how to construct geometry. For example, constructing an arc through 3 points is the same whether you are using BobCAD, AutoCAD, Unigraphics or whatever. However, if you are very familiar with other CAD programs, you may have to relearn how to do certain things.

BobCAD has a very powerful CAD engine. This means there are several ways to achieve any particular goal.

Like all CAD systems you can construct your geometry by entering absolute co-ordinates from the X,Y, Z origin on the screen. Similarly, you can create geometry using relative coordinates from a selected point.

BobCAD also gives you a large selection of tools for constructing various shapes such as arcs, rectangles, splines, surfaces, and so on. Starting with BobCAD v19, the software now offers a good range of tools for creating primitive solids such as cones, cubes, spheres, cylinders and tori.

If you do considerable surface creation, I strongly suggest you look at v20 BobCAD. The enhancements and upgrades from v19 to v20 are striking and powerful.

The User Coordinate System (UCS)

An important feature of BobCAD with which you will want to become familiar is the User Coordinate System or UCS.

When constructing your part, you may have features that are called out from existing features. *(See WCS 1)* In this example you can see the upper right square is located 1.00 in the X direction and .10 in the y direction from the center of the hole pattern.

If you were to place the square on the drawing using the absolute mode you would have to calculate the distance from the part origin to the center of the hole pattern and then add the additional offset to the lower right corner of the square.

BobCAD gives you a much easier way to solve that problem by allowing you to


WCS 1
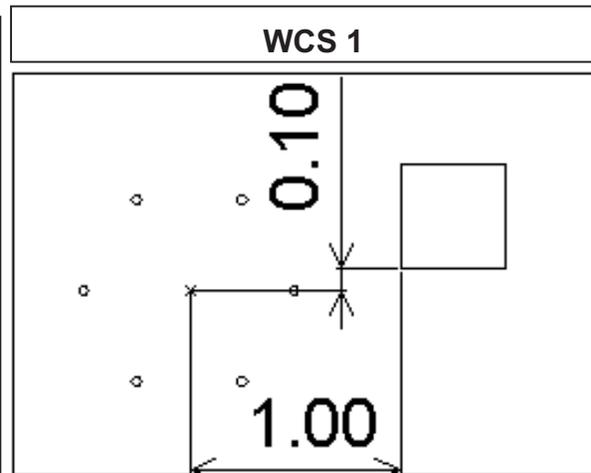
move the UCS. In the example above, I simply moved the UCS to the point in the center of the hole pattern and then defined my square. Below is a brief description of the steps involved, the details for each of the steps will be discussed in the following chapters:

1) Place a hole pattern on the screen
2) Place a point at the center of the hole pattern.
3) Right click on the BobCAD desktop to open the following menu.
4) Click on Modify Mode, shown highlighted in the picture.
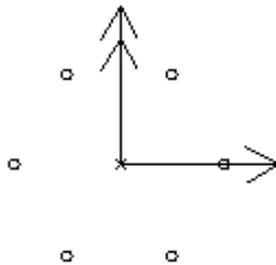5) Right click on one of arrows on the axis indicator to open the following menu
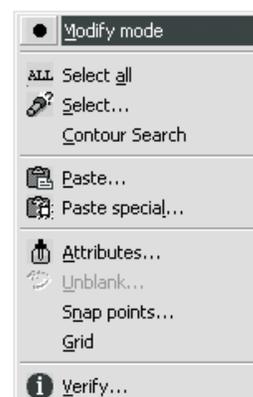
6) Click on the "Snap to" option. Notice the "P" to the right of "Snap to." That tells you that BobCAD will snap to an existing point.

↥	Sna_p to	P
⇆	Re_set	R
↻	_Rotate...	
⇲	Align _X	X
↻	_3D align	
⇱	Align to element	
↥	_Move...	
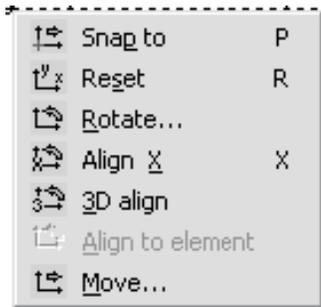
7) Click on the point in the center of the hole pattern. *(See result below)*

You can see the UCS has been moved to the point at the center of the hole pattern.
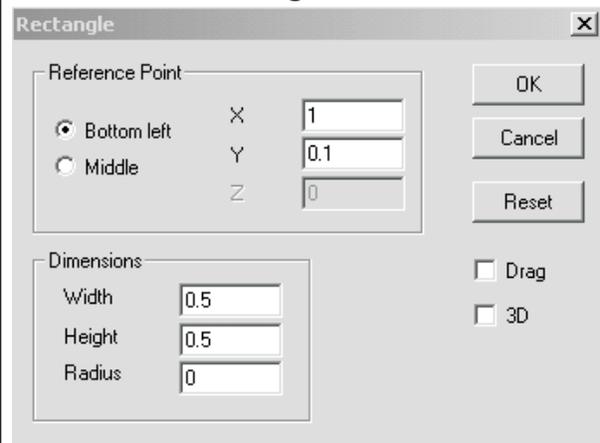
It's that simple.

The ability to "drag and drop" the UCS to any existing point means you can easily draw all features of your part in the most convenient orientation.

Not only can you drag and drop the UCS, you can also rotate it on the fly.

Rectangle Feature

Rectangle ☒

Reference Point
- ⦿ Bottom left
- ○ Middle

X [1]
Y [0.1]
Z [0]

OK
Cancel
Reset

Dimensions
Width [0.5]
Height [0.5]
Radius [0]

☐ Drag
☐ 3D

For example, imagine you are working on a work piece that will be programmed on three sides. You can quickly:

1) Move the UCS to the most convenient point on the project

2) Grab one leg of the axis and rotate it to an orientation that is more useful.

This is just an overview to get you familiar with the possibilities. We will cover this in later chapters and exercises.

Setting your Environment

There are many options available that enable you to customize your work space. For example:

- You can choose which tool bars you want to see.
- You can pick which icons you want on each tool bar.
- You can choose how the program uses snap points. (*If you are not familiar with this concept. Read the tool tip in the box on the next page.*)
- You can turn tool tips on or off. Tool tips are the balloon text that pops up when you

8) Open up the rectangle option off of the other menu and enter the distance to the square from the center of the hole pattern. *(See example)*

hover the mouse pointer over an icon.

The best way to learn about this is to:

A) Start up BobCAD.

B) Click on the File Menu.

C) Click on Environment to open the following dialog box.

D) Click on the various tabs across the top.

E) Experiment with the available settings on each tab and notice how your working environment changes.
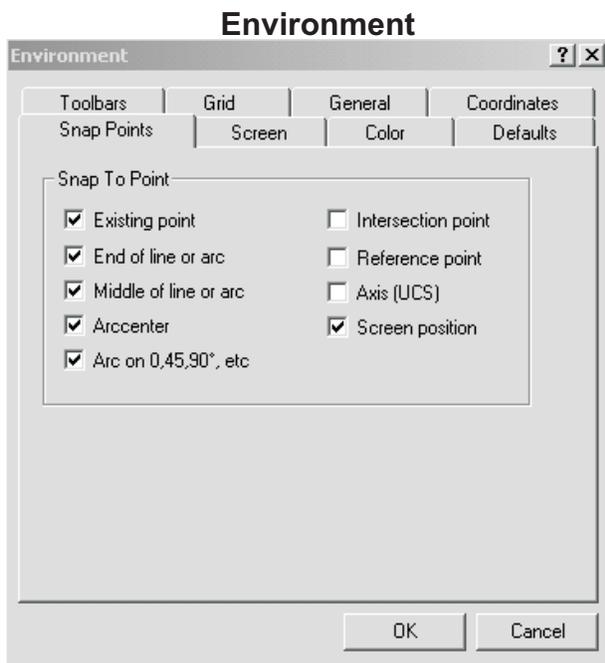
Environment

There will be times when you want to change the environment to fit the work you are doing. For example, you may not want to have the snap point set to screen position. Or, you may want a grid on the screen so you can use it to speed up drawing.

Another thing you will learn as you test the environment settings is that the basic unit of measure in BobCAD is metric.

Coordinate Settings
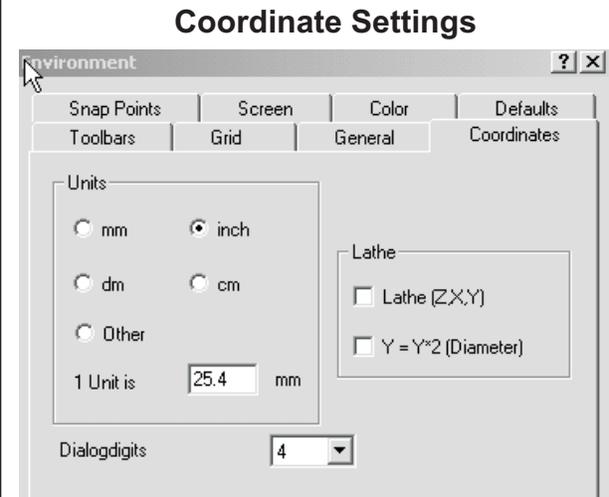
As you can see, you have your choice of using the inch or metric system.

There is a place on the screen where you can set up the coordinate system for lathe programming. And, you can choose whether you want to program the lathe using diametric or radius values.
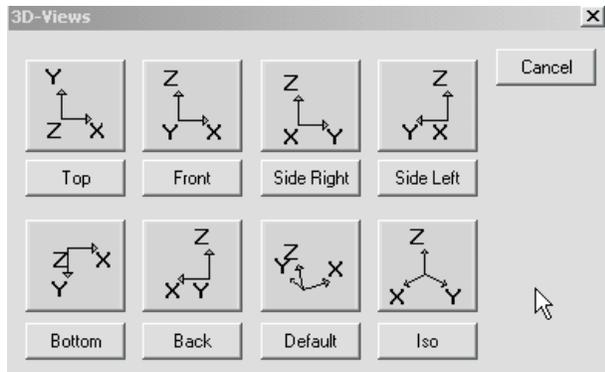
The Views

The Quickest way to change the way you view your project is to click on the icon in the upper right corner of the screen that looks like the picture shown here.
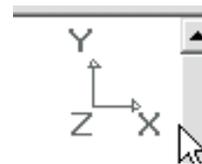
When you click on this icon, a menu will pop up (*see next page)* that allows you to select the view that is most appropriate for your immediate task at hand.

By clicking on any one of the buttons shown below, you can quickly change

from top view to side view, or from iso-metric to front view.

This is a very nice feature. Unlike some other systems where you have to locate the view menu, open up a drop down menu, then select the view you want, this feature makes it as easy as 2 mouse clicks.

If you prefer selecting your view from a menu. That feature is also available. Simply:

1) Click on the "Views" menu item.
2) Click on the "Select 3D View option.

You can see from the Views drop down

menu, you can do a lot more than just select 3D views. From this menu, you have the options to dynamically rotate the image, scale it, set the view options and so on.
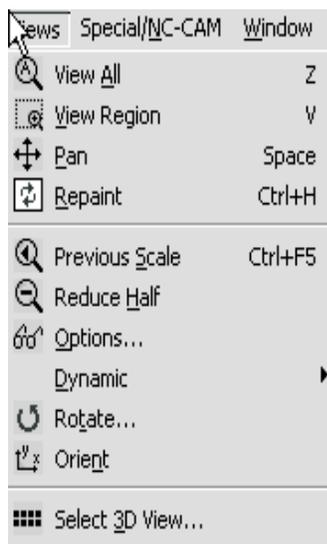
This is another menu that is well worth exploring. Later, when your

are working on the lessons, step through all of the viewing options and see how much power is available from a couple of mouse clicks.

Not so many years ago, this type of power was only available in the very expensive, high end programs.

Snap Points

To become efficient at BobCAD it is important to understand the concept of snap points.

You can set BobCAD so the cursor "snaps" into the desired location. For example, you may want to snap to arc centers only. In that case you would set the environment settings so that only the "Arc Center" option is checked.

Or, if you want to sketch in freehand mode, you would enable the "Screen Position" option.

Similarly you can choose to only snap to the ends of elements, or on arcs at specific angles, or at intersection points. This is particularly useful when working with complex drawings.

The more geometry you put on the screen, the easier it can be to snap on the wrong element. By selecting and changing your snap point options, your screen picks will be faster and more reliable.

Combine this with the ability to hide or reveal geometry based on color or layers and you have the tools to simplify and speed up the drawing process.