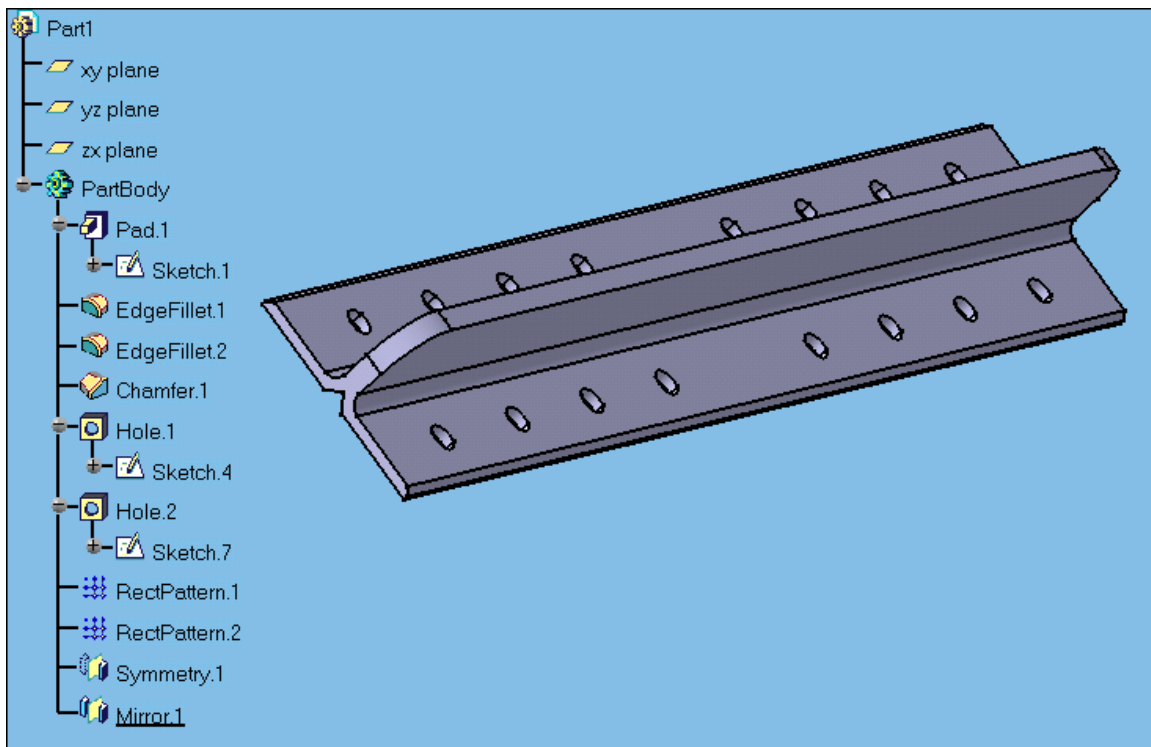


CATIA V5 Workbook

Release 3



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Introduction to the Sketcher Work Bench

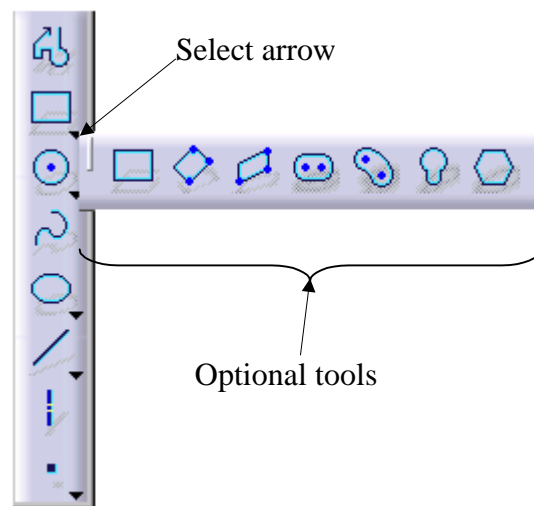
This lesson will take you through each step in creating a simple sketch and part that will be referred to as the “**L Shaped Extrusion**”. Later in this lesson you will be asked to save this part (file) as the “**L Shaped Extrusion.CATPart**”. The completed “**L Shaped Extrusion**” is illustrated in Figure 1.2. In some cases optional processes will be explained. Referenced illustrations will be used to help explain certain processes and to compare results. It is important that you complete and understand every step in this lesson, otherwise you will have difficulties in future lessons where much of the basic instruction will not be covered (it will be assumed that you know it). The concepts taught in these steps will give you the tools to navigate through the basics of the **Sketcher Work Bench**. Following the step-by-step instructions there are twenty questions to help you review the major concepts covered in this lesson. There are practice exercises at the end of this lesson. The practice exercises will help you strengthen and test your new found CATIA V5 knowledge. This lesson covers the most commonly used tools in the **Sketcher Work Bench**. The less common and/or advanced tools will be covered in later lessons and/or in the Advanced Workbook. It is not the intent of this book to be a comprehensive reference manual but provide basic instructions for the most common tools and functions in CATIA V5. CATIA V5 in the **Windows NT** environment allows multiple methods of accomplishing the same task. You are encouraged to explore all the different options.

Sketcher Work Bench Tool Bars

There are three standard tool bars found in the **Sketcher Work Bench**. The three tool bars are shown below. The individual tools found in each of the three tool bars are labeled to the right of the tool icon.

Some tools have an arrow located at the bottom right of the tool icon. The arrow ▼ is an indication that there is more than one variation of that particular type of tool. The tools that have more than one option are listed to the right of the default tool. To display the other tool options you must select and hold the left mouse button on the arrow as shown in Figure 1.1. This will bring up the optional tools

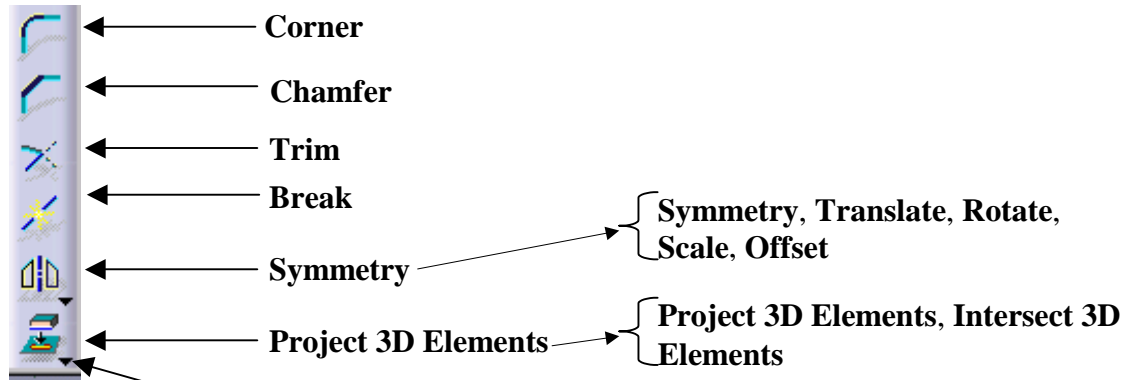
Figure 1.1



window. Move your mouse to the desired tool and release the mouse button. The desired tool icon now becomes the default tool, shown on the tool bar. All you have to do to select the new default tool is to double click on it.

The Operation Tool Bar

Tool Bar	Tool Name (default)	Tool Type Options
----------	---------------------	-------------------

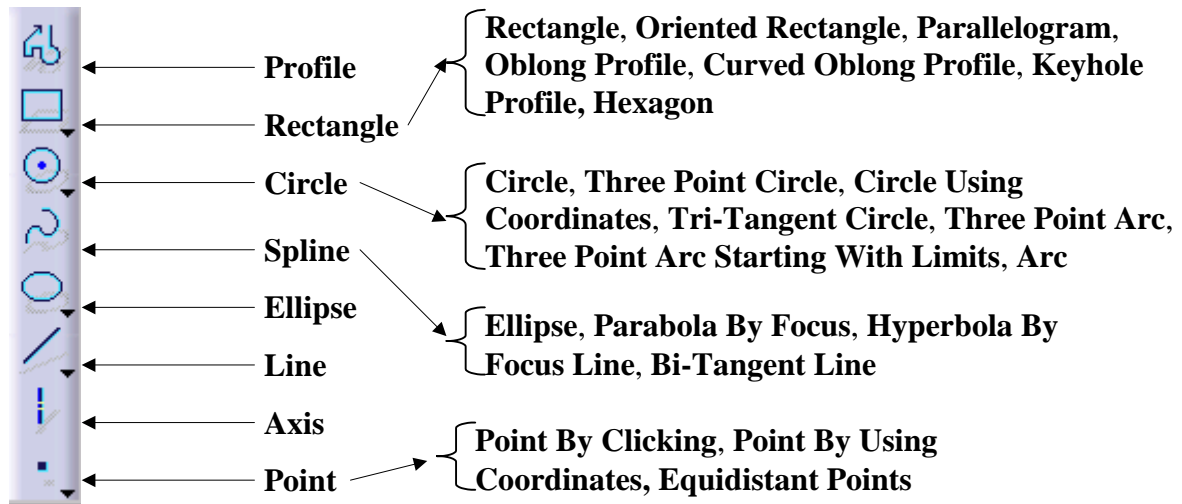


Note: Arrow indicates multiple tools are available. Click on the arrow and the other tool options will appear.

Tools covered in this lesson: **Corner, Chamfer, Trim and Break.** **Symmetry and Project 3D Elements** tools will be covered in Lesson 2.

The Profile Tool Bar

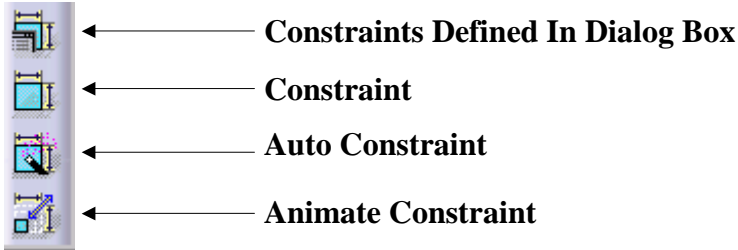
Tool Bar	Tool Name (default)	Tool Type Options
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Tools covered in this lesson: **Profile, Rectangle, Circle, Line and Point.**

The Constraints Tool Bar

Tool Bar **Tool Name (default)** **Tool Type Options** .



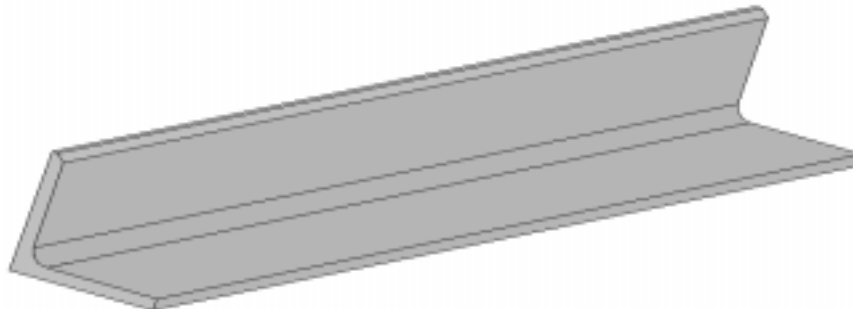
All of the constraint tools are covered in this lesson.

NOTE: The three tool bars are by default located on the right side of the screen. The three tool bars contain too many tools to show all of them at one time. To view and have access to all the tools you can select the shaded tab located at the top of each tool bar and drag it anywhere on the screen. This is important because when you get to Step 4, by the default setup you will not be able to visually locate the **Operation** tool bar. You will have to select and drag the **Operation** tool bar from the right bottom side of the screen to the location you select.

Steps To Creating A Simple Part Using The Sketcher Work Bench

You are now going to use the tools just introduced to you to create an “**L Shaped Extrusion**”. The part is referred to as an “**L Shaped Extrusion**” because it’s profile or shape is similar to an upper case letter L. When you complete all the steps in this lesson the result should look similar to Figure 1.2.

Figure 1.2



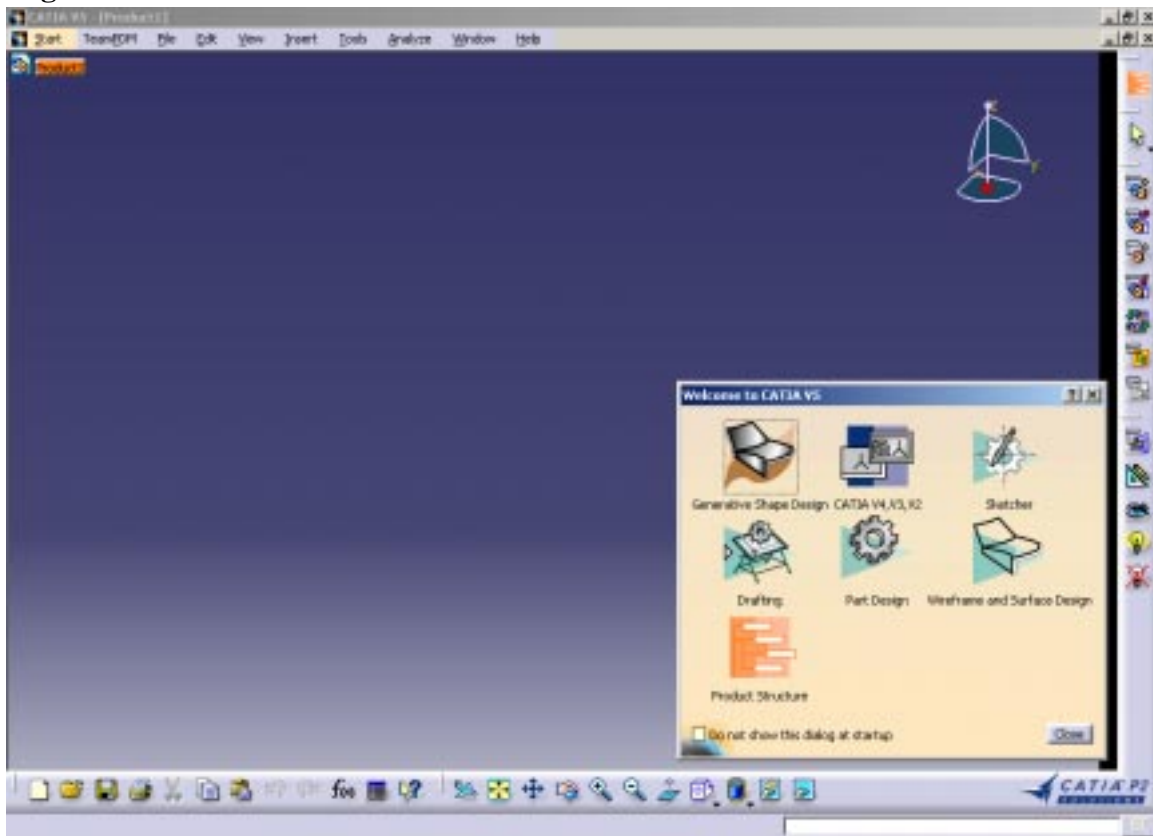
1. Start CATIA V5



From the **NT Desktop** double click on the **CATIA V5R3** icon. Be patient it may take a few moments to bring up the CATIA V5 start logo and the actual CATIA V5 working window. Figure 1.3 shows what the screen should look like.

If you are not able to finish all the steps in this lesson in one session you can jump to Step 23, which covers saving and exiting CATIA V5. This will allow you to save your work for your next session.


Figure 1.3



2 Select The Sketcher Work Bench.

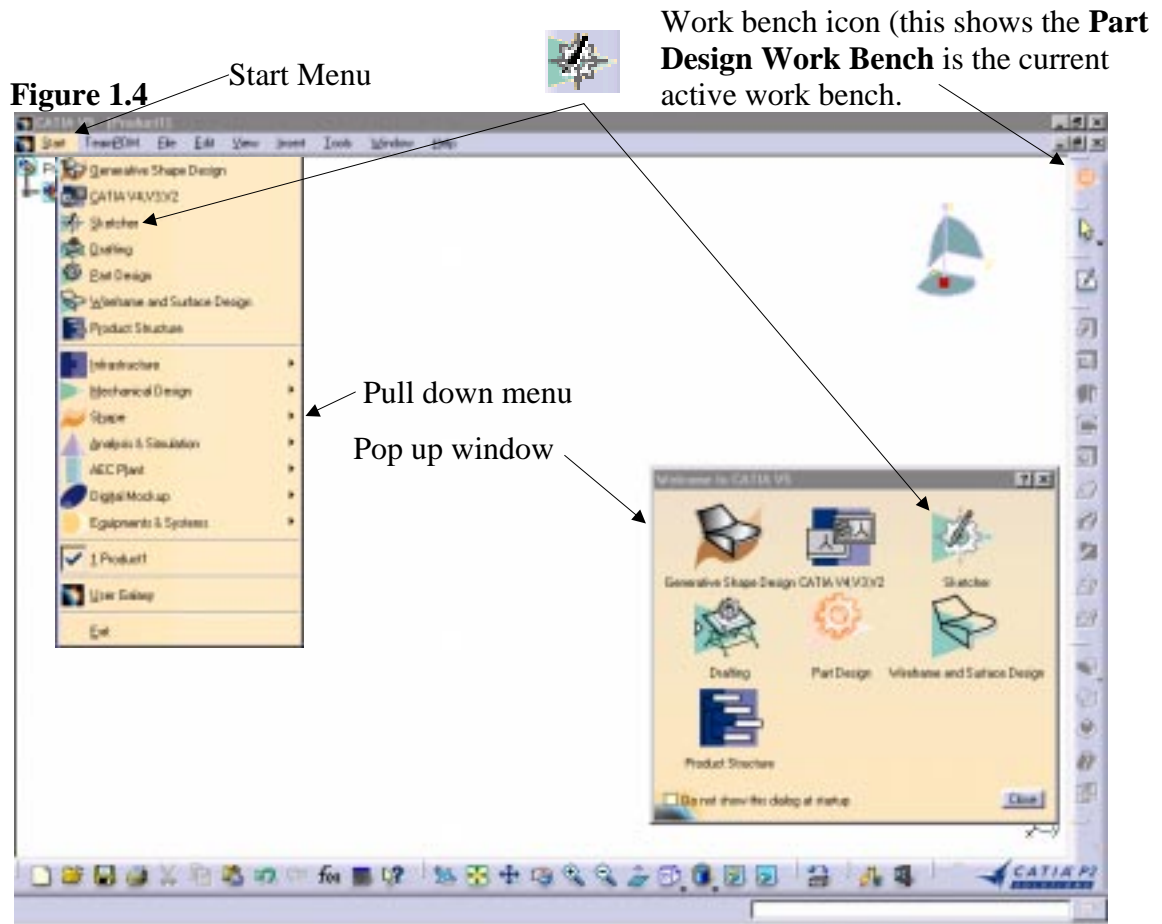


Every time you start CATIA V5 the CATIA V5 screen will appear as it does in Figure 1.3. The “**Welcome to CATIA V5**” pop up window will be prompting you to select a work bench. The default work bench is **Product Structure**. For this lesson you will need to select the **Sketcher Work Bench**. Notice as you select the **Sketcher Work Bench** that the tool bars on the right hand of your screen change and the “**Welcome to CATIA V5**” pop up window disappears. If your CATIA V5 screen and/or your **Sketcher Work Bench** screen are not maximized, maximize them using the windows function at the top right of the screen.

For future reference there are two methods to select a work bench in CATIA V5. As you start CATIA V5 you are prompted by the default method. Using the “**Welcome to CATIA V5**” pop up window is one way. Once you have selected a work bench and the “**Welcome to CATIA V5**” window has disappeared you can bring it back up by selecting the **Work Bench** icon  in the top right of your screen, reference Figure 1.4. The term work bench is used generically because the **Work Bench** icon showing will be the current active work bench. Selecting that work bench will bring up the “**Welcome to CATIA V5**” pop up window.

The other method of selecting another work bench is by selecting the **Start** icon in the top left side of the screen, reference Figure 1.4. This will bring up a pull down menu that includes all of the work benches. Double click on the work bench you want to use, in this case the **Sketcher Work Bench**.

Figure 1.4 shows what the menus look like on the screen for both methods described above. It is not possible to use both methods at the same time as shown in Figure 1.4 you can only use one method at a time.



NOTE: Selecting the **Work Bench** icon method will bring up the “**Welcome to CATIA V5**” pop up window. This window will contain only the default work benches at the time CATIA V5 was installed. This window can be customized. If your system has been customized your “**Welcome to CATIA V5**” window may have different work benches. The **Sketcher Work Bench** should be included in the default window.

3 Specify A Working Plane

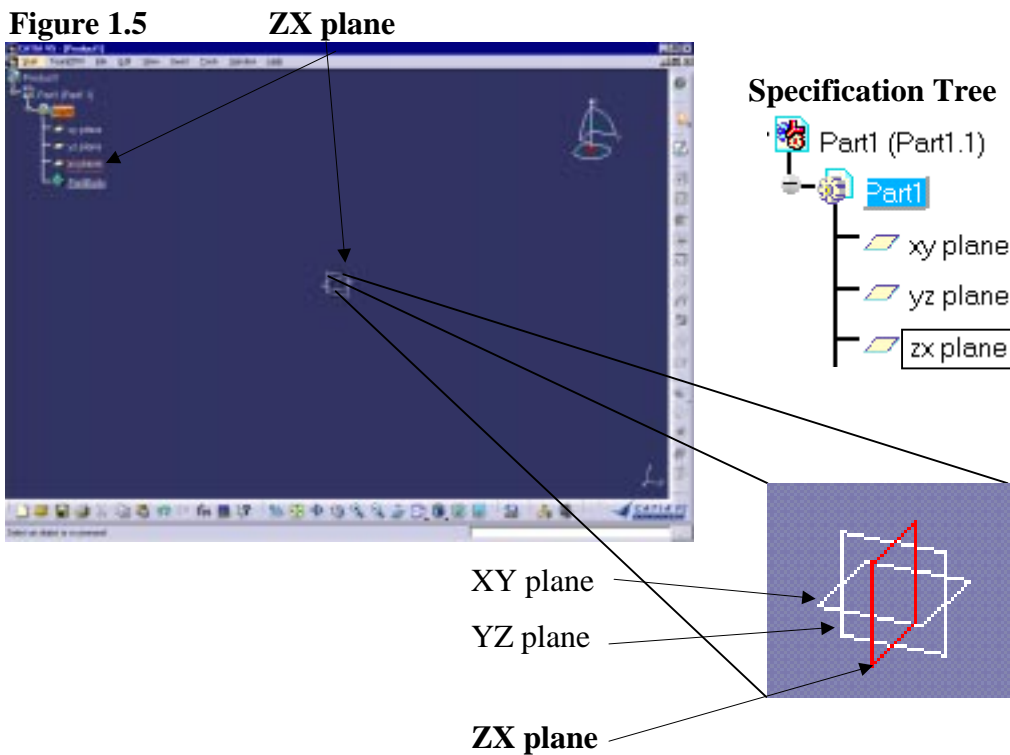
The next step is to create a 2 dimensional profile of the part. The **Sketcher Work Bench** is a two dimensional (planar) work area. To use the **Sketcher Work Bench** you must specify which plane the profile is to be created on. Specifying the plane can be done several different ways.

- 3.1 Select (highlight) the desired plane from the graphical representation in the center of the screen as shown in Figure 1.5. Notice as a particular plane is selected the equivalent plane in the **Specification Tree** is highlighted. If the **Specification Tree** isn't showing the branches with the

planes, it will need to be expanded. To do this just select the **Plus** symbol \oplus to the left of the **Specification Tree** or double click on the branch you want expanded.

- 3.2 The step described above can be reversed. Select the plane in the **Specification Tree** and the coordinating plane in the center of the screen will also be highlighted.
- 3.3 Other planes, surfaces and/or other planner objects can also be selected to define the **Sketcher** plane. This option will be covered in more detail later in the book.

For this lesson select the **ZX plane** as shown in Figure 1.5.



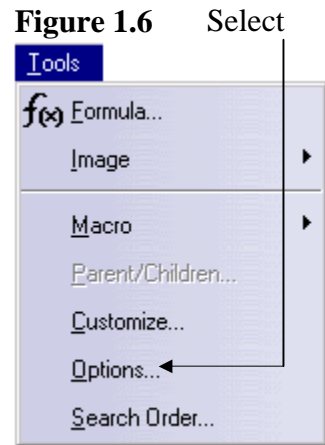
4 Entering the Sketcher Work Bench

Once a plane is selected the screen will animate, rotating until the selected plane is parallel to the computer screen (perpendicular to you, true size). The default grid will also appear. You are now officially in the **Sketcher Work Bench** but before you create the planar profile of the “**L Shaped Extrusion**”, you need to customize the grid.

NOTE: As mentioned in the introduction, CATIA V5 is Windows compliant. This means that there are several methods available to complete almost every task.

5 Customizing The Grid

5.1 Go to the top tool bar in the pull down menu and click on **T**ools, **O**ptions as shown in Figure 1.6. This brings up **file tab** options on the right side of the screen and **file type** options on the left (Figure 1.7). From the options on the left select **Part**, the tabbed options on the right change accordingly.



5.2 Select **Sketcher**. There are four main options under **Sketcher**; you only need to use two of them at this time, **Grid** and **Sketch Plane**.

5.3 The first option under Grid allows the user to select **Display** grid or not select it. For this particular exercise check the **Display** option.

5.4 The second option is to allow the user to snap to the grid points. For this particular exercise check the **Snap To** option.

5.5 The third option is **Primary Spacing**. The user can set the desired spacing. If the default measurement is in metric the spacing will be in mm. To change this default complete the following steps:

5.5.1 Select the **General** option on the left hand option bar. This is in the same window as described in Step 5.1 above.

5.5.2 Slide the File tab to the right till you find the **Units** tab, select it. The window on the screen should now look like Figure 1.8.

5.5.3 Highlight the **Length** option at the top of the list.

5.5.4 The length option will appear at the bottom of the window list.

5.5.5 Selecting the **down arrow** will give you a list of all the types of length measurements. For this exercise select **inches**.

5.5.6 Now go back to the **Sketcher** options, by selecting the **Part** option in the left window and selecting the **Sketcher** tab on the right. Notice the **Primary Spacing** option is now showing in inches.

Figure 1.7

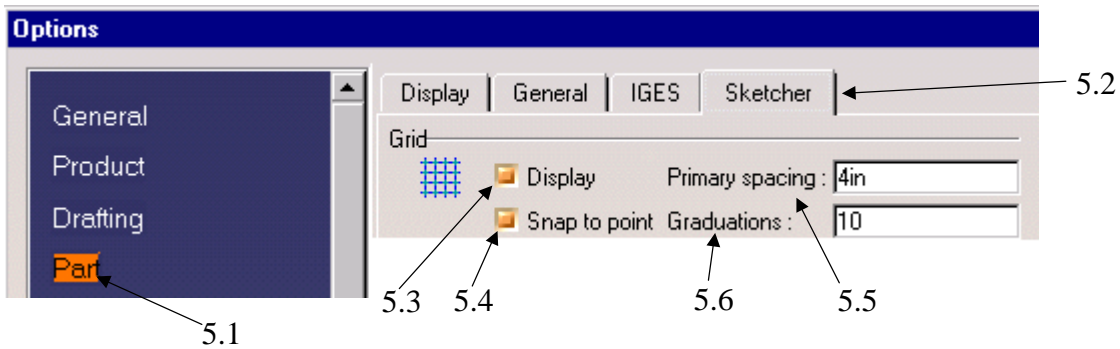
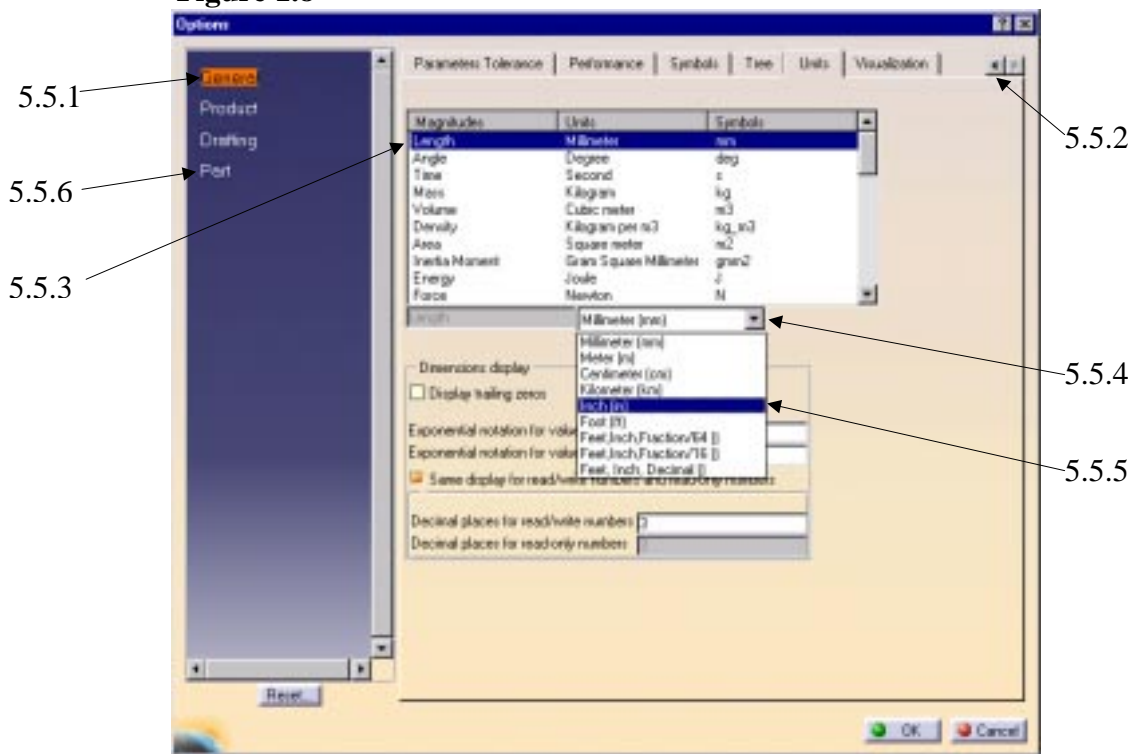


Figure 1.8



5.6 The fourth option under **Grid** is **Graduations**. This option divides the **Primary Spacing** in divisions defined by you, reference Figure 1.7. As an example if the **Primary Spacing** is 1” and the **Graduations** is 1 (division), the grid will remain in 1in grids. If the **Primary Spacing** is 1” and the **Graduations** set to 2 (divisions), the grid will be .5 in. To change the **Primary Spacing** and the **Graduations** just select the value in the window and type in the new value. When entering the values for the **Primary Spacing** it is not necessary to enter the measurement type. The lowest value allowed for **Graduations** is 1 (zero will not be accepted). For this exercise enter 1 for the **Primary Spacing** and enter 10 for the

Graduations. Select the **OK** button to apply the **Primary Spacing** and the **Graduations** values. The **Primary Spacing** is represented in the **Sketcher Work Bench** with a solid line while the **Graduations** is a dotted line (Figure 1.9). It is important to remember that the zoomed view on the screen will dictate how the **Primary Spacing** and **Graduations** are represented. If you are zoomed out, the **Graduations** and **Primary Spacing** could look very similar to each other, not distinguishable. If you find yourself in this situation use the **Zoom** tool on the tool bar at the bottom of the screen (Figure 1.10). Continue to zoom in until the **Primary Spacing** and **Graduations** are distinguishable.

Figure 1.9

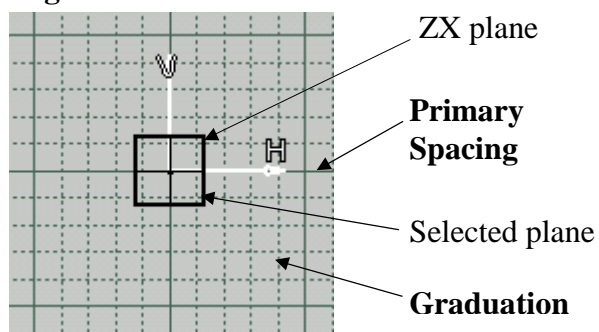
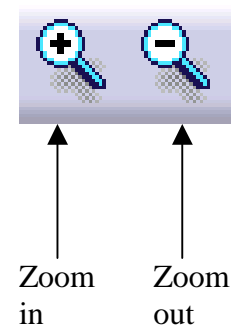








Figure 1.10



6 Creating Geometry Using The Profile Tools

You are now ready to create the profile (periphery) of the “**L Shaped Extrusion**”. The first tool you will use from the **Profile** tool bar is the **Point by Clicking** tool , covered in Step 7. The second tool is the **Line** tool , covered in Steps 8, 9 and 10. The third tool is the **Profile** tool , covered in Step 11. On the **Tools** tool bar at the bottom right of the screen make sure the **Snap To Point**  is on (highlighted), the **Geometrical Constraints**  is on and the **Dimensional Constraints**  is on (Figure 1.13). With this you are ready to create geometry!

7 The Starting Point

The (0,0) point in **Sketcher Work Bench** is the intersection of the **Horizontal (H)** and **Vertical (V)** axis. It can also be described as the intersection of the three planes (**XY**, **ZX** and **YZ**). Reference Figure 1.5, 1.9 and 1.12a.

The starting point for your profile will be (1,1). You should be able to locate the (1,1) location using the **Primary Spacing** and **Graduations**. To visually verify the location and to **Anchor** your first two lines to the (1,1) location create a point at the (1,1) coordinate location. To create a point complete the following steps:


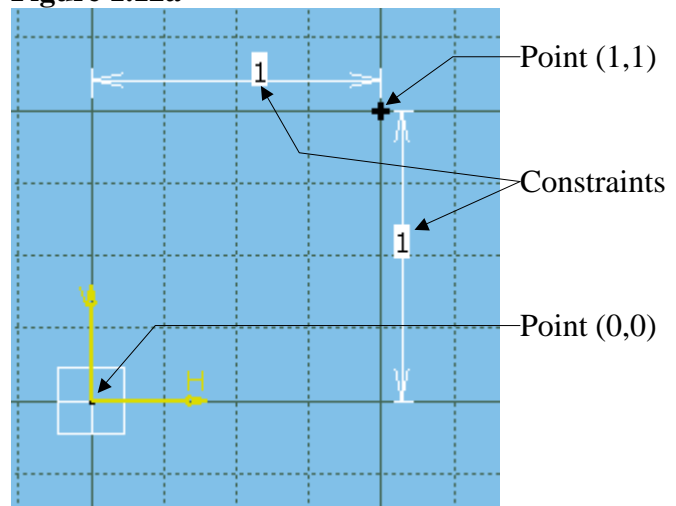
- 7.1 Select the **Point By Clicking** icon found in the **Profile** tool bar on the right side of the screen. After selecting the **Point By Clicking** icon the mouse will be accompanied by a **Target Selector**.  This tool allows you to select and snap to a location on the screen. CATIA V5 will prompt you to **“Click To Create The Point”**. Another way of specifying the location of the point is to type the location in the **Point Coordinates: H:** and **V:** boxes. The **H:** is for horizontal and **V:** is for vertical coordinates. Reference Figure 1.11.

Figure 1.11



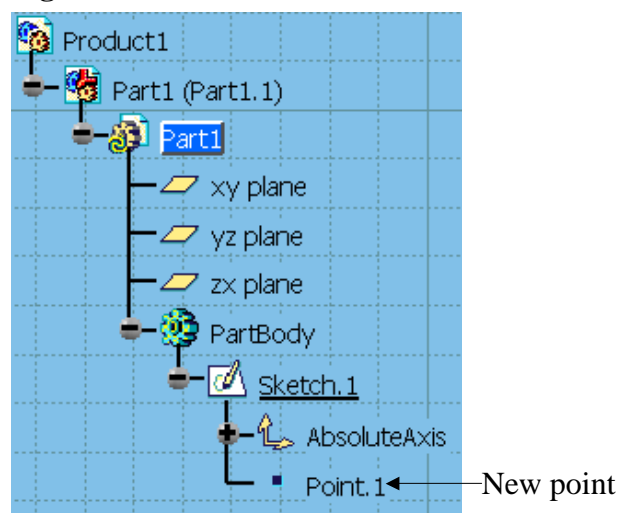
- 7.2 For this lesson type in 1 for the Horizontal coordinate. Hit the **Tab** key to move the cursor over to the Vertical box. Type in 1 for the Vertical coordinate. Hit the **Enter** key to have CATIA V5 create the new point.

Figure 1.12a



- 7.3 A Point “+” will appear at the (1,1) coordinate. It will remain highlighted until you make another selection. There will be two green dimension lines locating the point from the (0,0) location. The dimension values should be one in the horizontal direction and one in the vertical direction. The green dimension lines constrain the point to that coordinate location (Figure 1.12a). Notice a **Point.1** has been

Figure 1.12b



added to the **Specification Tree** (Figure 1.12b). Remember, you may have to expand the **Specification Tree** to see all the entities. **Point.1** will be under the **Sketch** branch.

8 Creating Line 1

Remember the grid you set up is 1in **Primary Spacing** with 10 **Graduations**. This means the dotted lines represents .1 of an inch. Complete the following steps to create line 1.



- 8.1 Select the **Line** icon from the **Profile** tool bar. This will bring up the **Tools** pop up window as shown in Figure 1.13. You will be prompted to **“Select A Point Or Click To Locate the Start Point”**. When you select the **Line** icon your mouse will be accompanied by a **Target Selector**. 

Figure 1.13

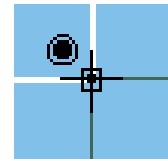


- 8.2 The starting point for line 1 will be **Point.1** created in Step 7. Using your mouse select Point.1 . You will now be prompted to **“Select A Point Or Click To Locate the End Point”**. The Tools pop up window will also update to prompt for the end point.
- 8.3 The end point for line 1 is (1,2). If you can use the grid to locate the correct location do so. Move your **Target Selector** up one full grid line but don't move it to the right or left (0 in the horizontal direction). Click on the grid line intersection (1,2). If you have any doubt where (1,2) is type in the values, using the **Tools** pop up window. Type in 1 for the **H:** box and 2 for the **V:** box.
- 8.4 The first line is now created. Line 1 should look like the one labeled in Figure 1.15.

Notice: Connecting one entity to another is safer and easier when the **Snap To Point** icon  is on. When the **Snap To Point** icon is off you must be careful when connecting one entity to another. Both entities must share the same common point. For example, two connected lines, the end point for the first line must be the same exact starting point for the second line. The lack of a shared point will make the entities unlinked. This broken link will cause problems when moving and/or modifying your profile. The entities will not move together. Another problem with the broken link is that it creates an unclosed profile. Unclosed profiles will be covered later in this lesson.

CATIA V5 does supply a visual tool to help you know exactly when the point being selected is shared with another entity. The symbol is shown in Figure 1.14, the blue circle filled with a blue dot signifies the point being selected is the end point of another entity. This will link the two entities together. This is a helpful tool, especially when the **Snap To Grid** tool is off.

Figure 1.14

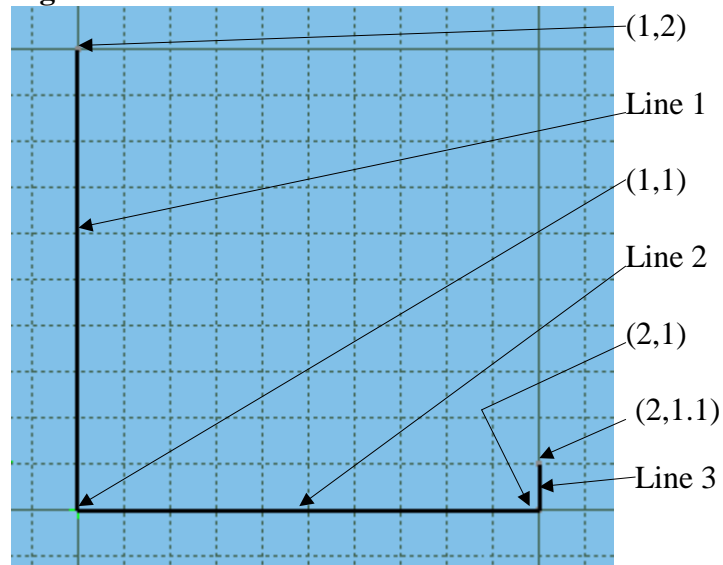


Notice: The Tools pop up window gives you more options than the ones covered in Step 8.1 & 8.3. If you are typing in the information to create a line you have the option of giving Polar Coordinate information. Reference Figure 1.13, you enter a **Start Point**, **L:** (length of line) and **A:** (for angle). This lesson does not require you to use this option, it could be helpful in the future.

9 Creating Line 2



To create the second line you have to re-select the **Line** icon. Repeat the same process described in Steps 8, except use (1,1) as the **Start Point** and (2,1) as the **Ending Point**. This will create the bottom horizontal line as shown in Figure 1.15.

Figure 1.15



10 Creating Line 3

To create the third line, double click on the **Line** icon. Double clicking on the **Line** icon will allow you to create multiple lines without being required to repeatedly select the **Line** icon. With the **Line** icon double clicked, create line 3, **Start Point** (2,1). The **End Point** for line 3 is (2,1.1). Double clicking on the **Line** icon still requires you to select a **Start Point** and an **End Point** every time, but you will not have select the **Line** icon for every line.

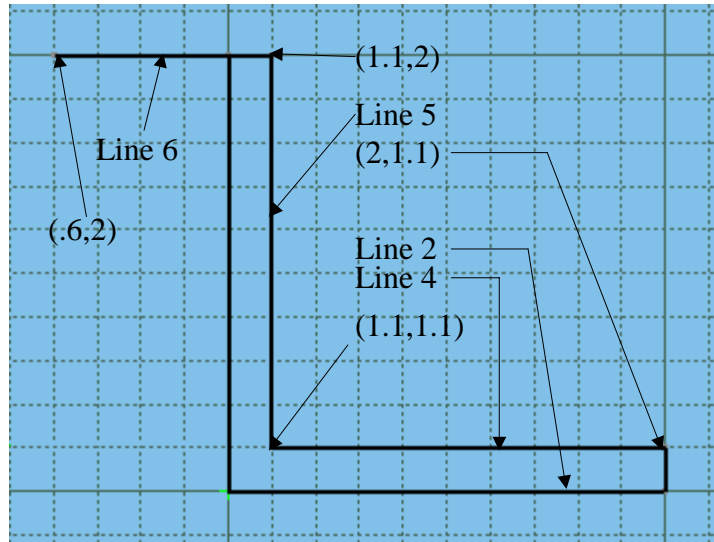
Note: If you make a mistake when creating one of the lines you can use the **Undo** icon.  The **Undo** icon is located at the bottom of the screen. The **Undo** tool allows you to undo multiple steps. Another option to a mistake is deleting it. This can be done using the **Cut** icon  also located at the bottom of the screen. Highlight the entity to be deleted then select the **Cut** icon.

11 Creating Line 4, 5, And 6 Using The Profile Icon

The 4th, 5th and 6th line will be created using the **Profile** icon. The **Profile** icon allows true successive line creation. The **End Point** for one line and the **Start Point** for the next line requires only one selection. The connected lines will continue to be created

with every point selected until you double click. Double clicking the **Ending Point** will end the **Profile** command. The lines created are separate entities, but the command that created them is recognized as one, so if you select the **Undo** command all the lines created in one **Profile** operation will be undone. With this tool added to your toolbox of knowledge finish the “**L Shaped Extrusion**”. Create lines 4, 5 and 6 by selecting the following coordinates in succession, select (2,1.1), select (1.1, 1.1), select (1.1,2) and double click on (.6, 2) to end the line creation. The finished profile should look like Figure 1.16.

Figure 1.16



NOTE: This particular exercise does not require any features with radii but the **Profile** tool has the ability to create them. Instead of selecting an **End Point** and a **Starting Point** for line creation, select the point (where the arc is to begin), hold down the left mouse button and drag it away from the starting point, then release the mouse button. You will notice as you drag the mouse button around the arc radius and location change. Move the mouse around to where you get the radius you want then select that point on the screen.

Steps 12 through 16 give instruction on how to use additional tools to modify the entities you have created.

12 Breaking Line 6

Step 11 purposely instructed you to create line 6 longer than required. In this step you will learn how to break a line. Step 13 will instruct you on how to trim line 6 back to line1. To break line 6, simply select the **Break** icon from the **Operation** tool bar. Select line 6 as shown in Figure 1.17. The line will highlight then select a location on the line where you want the line broken. For the purpose of this lesson select approximately three **Graduation** lines from the left end point (Figure 1.17). The line is now broken. The easiest way to verify this is to select the broken line, only one of the two line segments will highlight. You could also select the **Measure**

icon found at the bottom of the screen (Figure 1.18). Select the Measure icon then select (apply to) the line you want to measure. This would tell you how long the selected (broken) line is.

Figure 1.17 (trimming line 6)

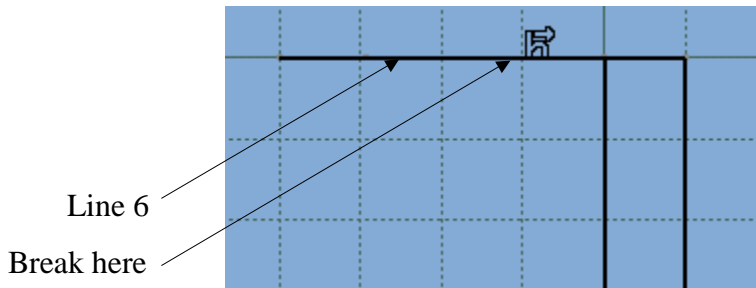


Figure 1.18 Measure icon

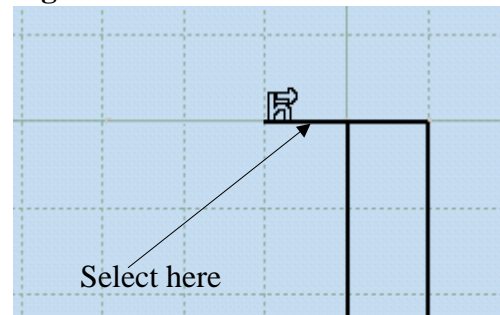


13 Deleting The Broken Line



This is another easy step but one that should be remembered. Select the left line fragment of the former line (known as line 6). It will highlight, now select the scissors located at the bottom left of the screen. The highlighted line will disappear (Figure 1.19). You could also select the **C**ut command from the top pull down menu (under **E**dit) or hit the **D**elete key. This deleting (erase) process is similar in all windows functions and applies to any entity you want to delete (as long as it is highlighted).

Figure 1.19



14 Completing The Profile Using The Trim Icon



The periphery of the “**L Shaped Extrusion**” is now complete, or is it? Extending line 6 past line 1 does not close the profile properly. If you were to exit **Sketcher Work Bench** at this point and try to extrude the profile you would get an error, because line 6 is over running line 1. To fix this problem select the **T**rim icon and select line 6 on the right side of line 1. Now select line 1, line 6 is automatically trimmed to the second line selected. See Figure 1.20 for line selection and Figure 1.21 for final result, after trim.

Figure 1.20

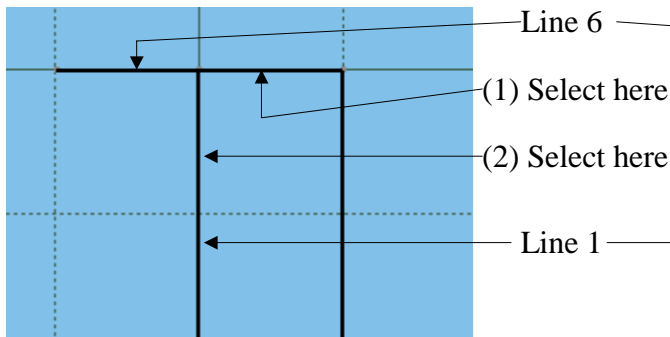
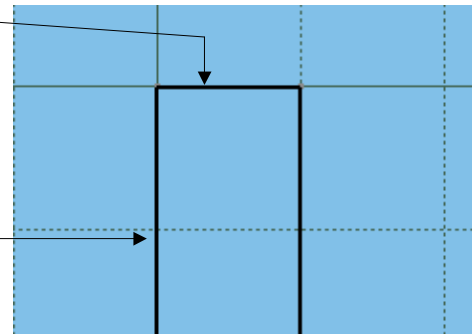


Figure 1.21 (line six after trim)



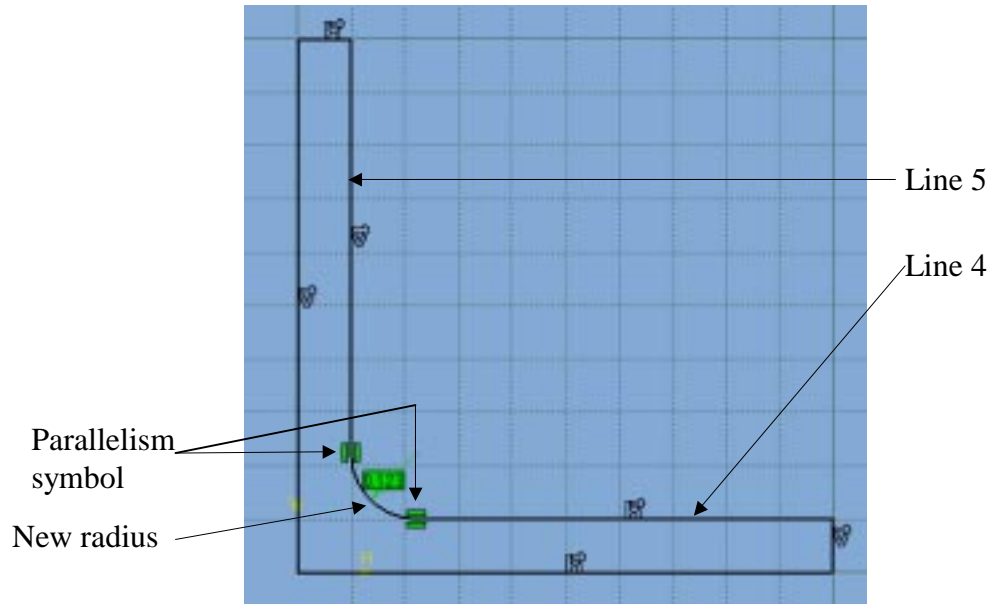
15 Modifying The Profile Using Corner



The **Corner** icon is located in the **Operations** tool bar. This tool modifies existing entities; in this case it will put a specified radius in the place of a square corner. The following instructions step you through the process of creating corners (fillets).

- 15.1 Select the **Corner** icon.
- 15.2 The command prompt at the bottom left hand of the screen will prompt you with the following: **“Select the first curve, or a common point”**.
- 15.3 For this exercise select line 4 (Figure 1.22).
- 15.4 The next command prompt will ask you to **“Select the second curve”**.
- 15.5 For this exercise select line 5 (Figure 1.22).
- 15.6 Now move your mouse around, the radius of the corner you just created will grow and shrink according to the location of your mouse. The command prompt will prompt you to **“Click to locate the corner”**, in other words move the mouse until the radius of the corner is where you want it and click.
- 15.7 You now have a radius for that corner. Your part should now look similar to the part shown in Figure 1.22. If your radius dimension does not match the one shown below it is ok, it will be modified later.

Figure 1.22 (sketch with radius added)



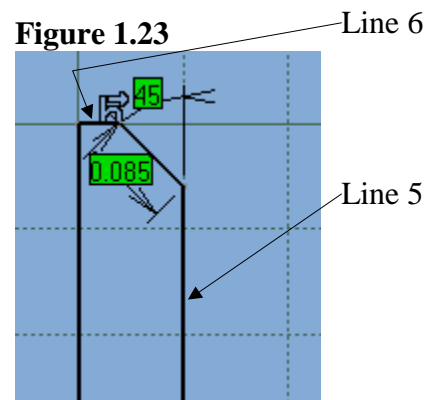
NOTE: The radius will have a green dimension with a value attached to it. The value is the radius of the corner you just created. Step 19 (modifying constraints) will supply us with the tools to make this radius exact. This is a two dimensional corner. Lesson 2 will explain another method of creating a corner using a **Part Design Work Bench**.

16 Modifying The Profile Using Chamfer



The **Chamfer** icon is also located in the **Operations** tool bar. This procedure assumes you know what a chamfer is. The steps required to create a chamfer are almost identical to creating a corner.

- 16.1 Select the **Chamfer** icon (shown above).
- 16.2 The command prompt at the bottom left hand of the screen will prompt you with the following: **“Select the first curve, or a common point”**.
- 16.3 For this exercise select line 5 (Figure 1.23).
- 16.4 The next command prompt will ask you to **“Select the second curve”**.



- 16.5 For this exercise select line 6 (Figure 1.23).
- 16.6 Now move your mouse around, the length of the chamfer will grow as you move the mouse away from the intersection of the two selected lines. The length of the chamfer will shrink as you move it back towards the intersection. If you move the mouse to the top left quadrant you will notice the chamfer also moves to that quadrant. CATIA V5 gives you the option of all four quadrants. For this lesson use the bottom left quadrant. The command prompt will prompt you to **“Click to locate the chamfer”**.
- 16.7 You should now have a chamfer that looks like the one shown in Figure 1.23.


NOTE: The chamfer has two green colored dimensions attached to it. Both dimensions have values attached to them. One dimension is the chamfer length and the other is the chamfer angle. Reference Step 19 (modifying constraints) on how to modify the values to exactly what you require for your chamfer. This chamfer is a two dimensional entity. Lesson 2 also explains a method of creating chamfers on three-dimensional entities, using a **Part Design Work Bench**.

17 Anchoring The Profile

Select line six. As you select the line hold the mouse button down, now drag the mouse up. Notice that the entire profile expands and contracts as you drag the mouse button around. Line 1 and 2 can be modified in length only, they can't be moved. All the other lines can be modified in position, length and angle. You cannot modify the location of lines 1 and 2 because they are linked to **Point.1** and **Point.1** is constrained to the location (1,1). The green dimension lines that were created with **Point.1** are constraints. It is the constraint values that tie **Point.1**, line 1 and 2 to their current position. To move the point and/or either line you have to modify the constraint, which will be covered in Step 19.

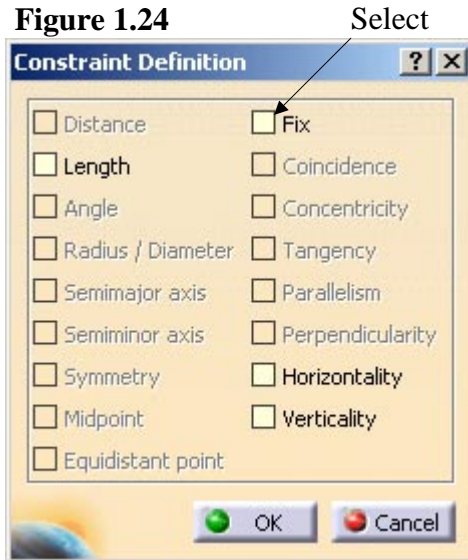
If there is a particular entity you don't want moved in relationship to another entity you can constrain it. Constraints are restrictions on one entity to another entity. The **Anchor** tool restricts the entities movement in relationship to the coordinate location only. Line 1 and 2 are not truly anchored because the constraint is tied to their relationship to **Point.1**. The effect is the same, line 1 and 2 can not be moved. If you want to constrain the location of an entity without constraining any other entity the **Anchor** tool is a good option. For example, you may want to modify the **“L Shaped Extrusion”** but you know you don't want line 6 to move at all. You can restrict line 6 by Anchoring it. Elements can be anchored by completing the following steps.

- 17.1 Select the entity that you want to anchor. For this lesson select line 6.

17.2 Select the **Constraints Defined In Dialog Box** icon . This will bring up the **Constraint Definition** pop up window. Reference Figure 1.24.

17.3 The **Constraint Definition** pop up window gives you a lot of options as far as selecting a constraint. For this lesson select the **Fix** constraint.

17.4 Select the **OK** button to apply the **Fix** constraint. Notice that line 6 will turn green meaning that it is constrained and the **Anchor** icon also shows up on the line, this signifies what kind of constraint is applied (Figure 1.25).

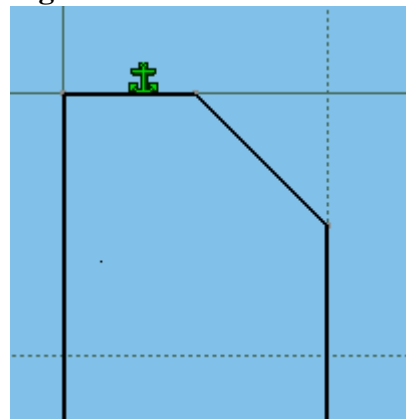


Allowing the quick and sometimes uncontrolled modification to a sketch can be a powerful tool, especially in the beginning stages of a design. As the design nears completion the ideas are being locked down, there are fewer variables. This is where CATIA V5 constraints come to the aid of the designer. As variables become known constants you can constrain them.

The purpose of this step was to give you a brief introduction to how CATIA V5 allows you to move and modify the sketched entities. It also introduces you to how to constrain the entities.

The only way to fully understand all the tools available to you is to test them yourself. Step 18 covers constraints in more detail.

Figure 1.25



18 Constraining The Profile

There are several reasons why you would want to constrain your profile. One reason is that you or any one else could accidentally select a line and move it out of position, as you experienced in Step 17. Constraints keep the required relationships between the **Sketcher** entities that make up the profile. There are multiple ways of constraining a part in CATIA V5. The nice thing about CATIA V5, constraining is optional, not required. Hopefully this step will convince you that constraints can be a powerful tool.

18.1 Constraint



This tool allows you to create individual constraints, one at a time. You have already applied a constraint and may not even know it. The **Anchor** icon in Step 17 is a constraint. The values attached to the **Chamfer** and **Corner** are constraints. To apply **Dimensional Constraints** complete the following steps:

18.1.1 Select the **Constraint** icon.



18.1.2 Select the line and/or **Sketcher** element to be constrained.

18.1.3 The **Sketcher** element will turn green (constraint symbol) along with the appropriate dimension and box with the value in it.

18.1.4 To re-locate the constraint value, select the value box and drag the mouse to the desired location.

18.1.5 If the initial location of the constraint is not satisfactory re-select the dimension and drag and drop it at the new location.

18.1.6 To edit the value of the constraint double click on the value box. This will bring up the **Constraint Definition** pop up window shown in Figure 1.26. This window shows the existing value for the **Sketcher** element. This value can be edited by typing the new value over the existing value. Then select **OK** or hit the **Enter** key. The entity linked to the constraint will automatically be updated to the new value.

If the constraint is between two different entities, such as lines, select the first line and then the second line. CATIA V5 will constrain the distance between the two entities. The constraint value will appear near the constraint. To move the

constraint value, follow Steps 18.1.4 and 18.1.5. For this lesson constrain your **“L Shaped Extrusion”** similar to the one shown in Figure 1.27.

Figure 1.26

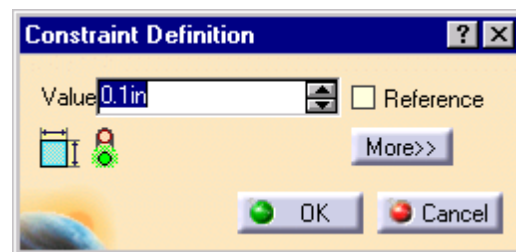
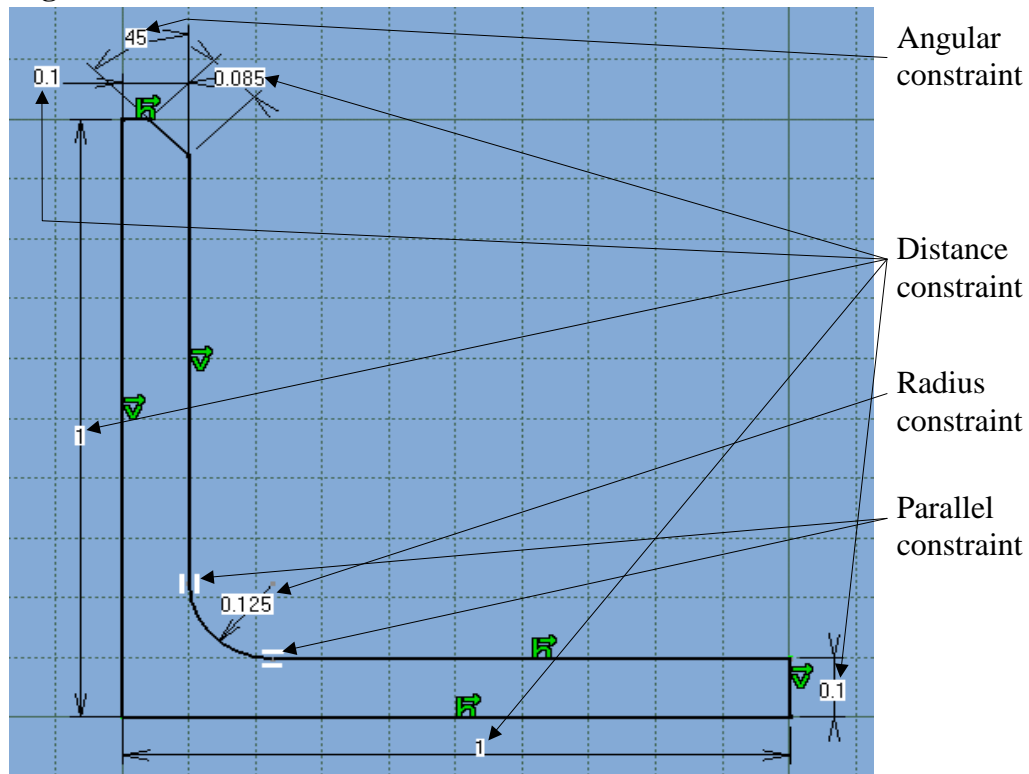


Figure 1.27

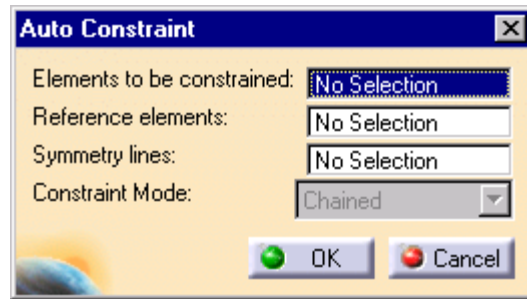


18.2 Auto Constraining The Profile



This method accomplishes the same task as the **Constraint** tool just explained, except that **Auto Constraint** can be much quicker (automatic). Once you select the **Auto Constraint** icon a pop up window comes up prompting you to select which entities you want to constrain (Figure 1.28). You can select one entity at a time, multi-select or select only a few specific entities that you want constrained. After making your selection select **OK**, located at the bottom of the pop up window. The entities selected will show up in green with the constraint value box. Getting complete control of this tool will take some practice and patience. If you feel brave use this tool to constrain your “**L Shaped Extrusion**” and see if you get the same result shown in Figure 1.27.

Figure 1.28

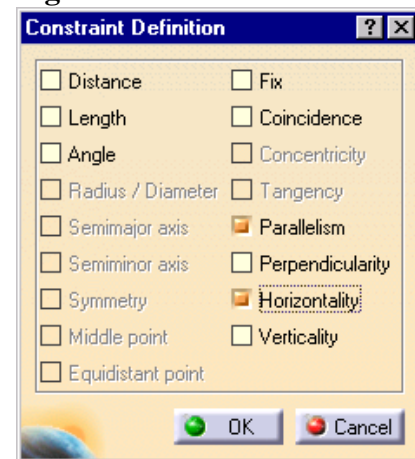


18.3 Constraint Defined In Dialog Box



To use this tool you have to select one or more entities and then select the **Constraint Definition In a Dialog Box** tool. A **Constraints Definition** box will pop up (Figure 1.29). The box will contain all the possible constraints but not all will be selectable. The only selectable constraints are the ones that apply to the entities selected. For example, if you selected one line you could apply the **Length**, **Fix** and **Horizontality** constraints, all the other constraints will be dimmed (meaning they are not selectable). CATIA V5 will not allow you to select the **Radius/Diameter** constraint because it does not apply to lines. Relationships between entities can also be established using this tool. For example, if you wanted **Parallelism** and **Horizontality** constraints between the top profile line and the bottom profile line on the base leg of the “**L Shaped Extrusion**” you would do the following:

Figure 1.29



18.3.1 Select both the bottom and top line of the base leg of the “**L Shaped Extrusion**” (lines 2 and 4 shown in Figure 1.30). This is a windows multi-select task, which is accomplished by, holding down the **CTRL** key while selecting both lines. Both lines will highlight.

18.3.2 Select the **Constraints Defined In Dialog Box** icon.

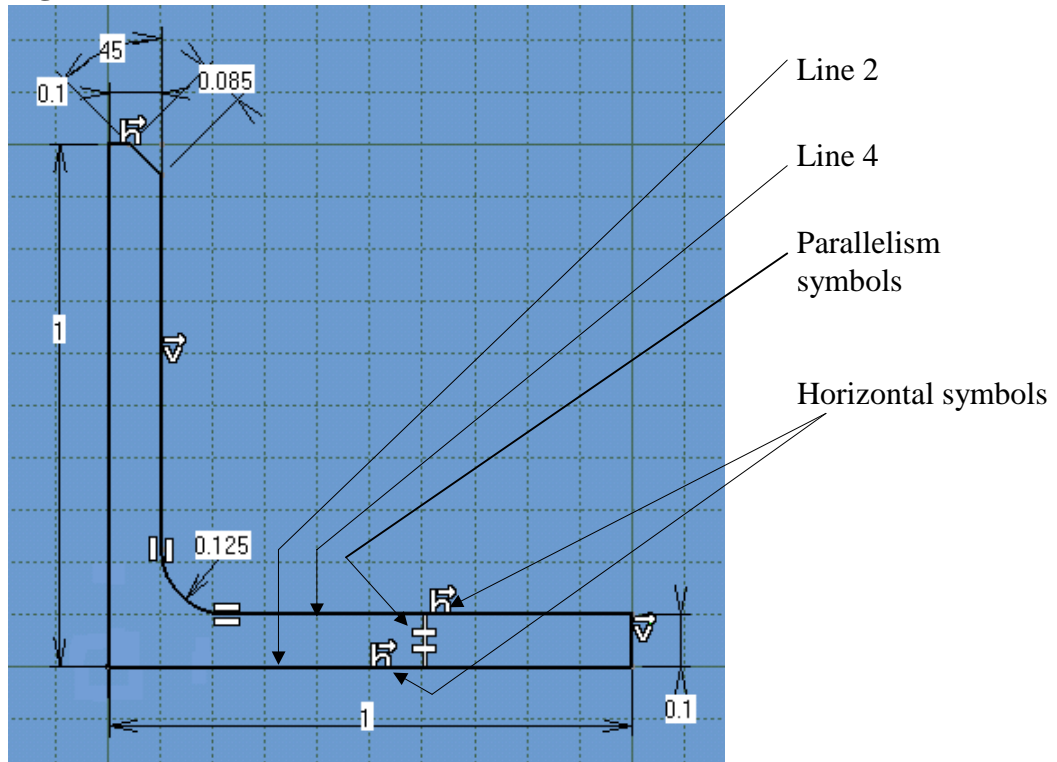
18.3.3 The **Constraints Definition** window will pop up (Figure 1.29).

18.3.4 Select the **Parallelism** box and the **Horizontality** box.

18.3.5 Select **OK**.

NOTE: The constraints that appear on the sketch are, the **Parallelism** and **Horizontality** symbols reference Figure 1.30.

Figure 1.30



The only way to really get complete control of this tool is to use it, experience it, and don't be afraid to make a few mistakes (that's why there is the **Undo** button).

18.4 Animate Constraint



The **Animate Constraint** tool allows you to visualize the effect one constraint has on the entire profile. This is a very helpful tool but be aware you may not always end up with what you started with. Remember, entities will not always stay attached as other entity values change. CATIA V5 will remember the relationships the different entities have with each other, if they were created with a relationship. For example, if the end point of one line is the same as the start point of another line it does not mean there is any relationship between the two lines. To use this tool follow the steps listed below:

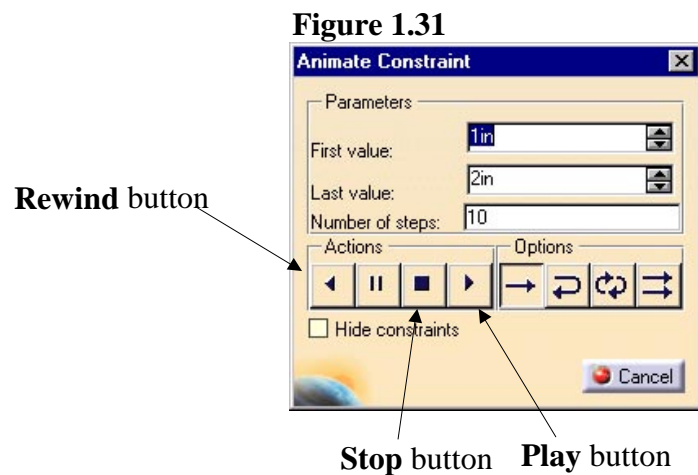
18.4.1 Select one existing constraint, only one constraint can be animated at one time.

18.4.2 Select the **Animate Constraint** icon



- 18.4.3 The **Animate Constraint** window will pop up (Figure 1.31).
- 18.4.4 Modify the parameters as desired/required and/or accept the default values.
- 18.4.5 Select the **Play** button. This will start the animation from the starting limit to the ending limit.
- 18.4.6 Watch the profile change as the selected entity animates from the first value to the last value. The **Animate Constraint** window has other options that you can test.

Notice: If your profile has entities created without relationship to other entities the **Rewind** button could result in a different profile than what you started with.



Animate Constraint is a powerful tool. It can help you visualize the change. It allows you to visualize without committing to a particular value.

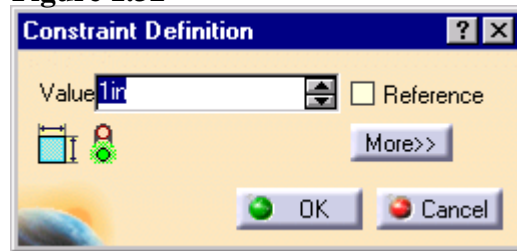
19 Modifying The Constraints

This process was previously described in Step 18.1.6. The ability to modify constraints in CATIA V5 is essential so the following steps are for your review.

- 19.1 Select the value box of the constraint you want to modify.
- 19.2 The **Constraint Definition** window will pop up (Figure 1.32). This window shows the existing value for the **Sketcher** element.
- 19.3 Edit the value by typing over the existing value.

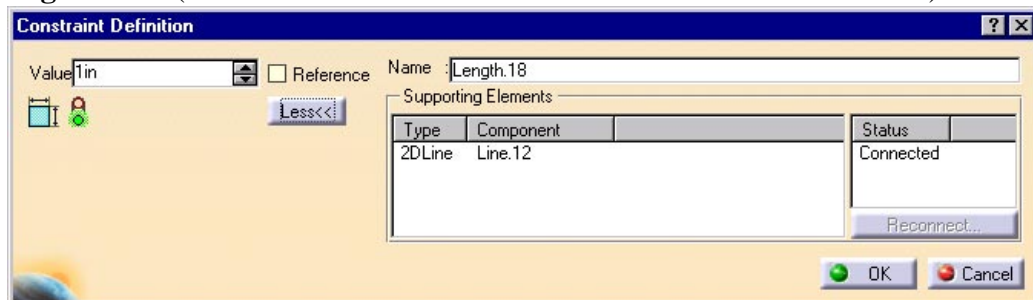
- 19.4 Apply the new value by selecting the **OK** button or pushing the **Enter** key.
- 19.5 The entity linked to the constraint will automatically be updated to the new value. Your profile updates automatically.

Figure 1.32



If you want to know more information about a particular constraint, double click on it and the **Constraint Definition** window will pop up. Select the **More** button to get detailed constraint information. Figure 1.33 shows how the **Constraint Definition** window looks when the **More** button is selected.

Figure 1.33 (Constraint Definition box with the More button selected)



Let's see what you can learn about one of your constraints on the **"L Shaped Extrusion"**. Double click on the constraint on the bottom line of the base leg. From the **Constraint Definition** window select the **More** button. The pop up window gives you information on other entities the selected constraint is connected (linked) to. It gives you the opportunity to change the name of the constraint that shows up on the **Specification Tree**.

20 Over Constraining The Profile... Not A Good Thing !

It is possible to over constrain a profile in **Sketcher Work Bench**. When you over constrain the profile CATIA V5 will inform you that you have a problem. CATIA V5 definition of over constraining is putting two different constraints on

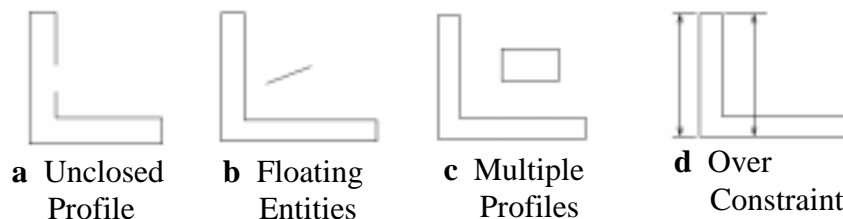
one or more entities. The two constraints can be correct individually but collectively have conflicting values. When an over constrained condition exists CATIA V5 will turn all the affected constraining values purple. Purple is the default color for over constrained sketches. Remember an over constraint condition is not a good thing. CATIA V5 will not allow you to extrude an over constrained profile. The easiest way to get out of the over constrained condition is to **Undo** or **Cut** the last constraint created, the constraint that caused the over constrained condition. You must reconsider which constraints are necessary to accomplish what you want. In the case of the **“L Shaped Extrusion”** you are creating the constraints that are used to maintain the specified dimensions. If your profile is not over constrained, you are ready to move on to the next step. If the instructions were followed an over constrained condition will not exist.

21 Exiting The Sketcher Work Bench


If your **“L Shaped Extrusion”** is similar to the one shown in Figure 1.27 you are ready to move the profile into the 3D world, the **Part Design Work Bench**. As a reminder the following conditions will not allow you to successfully extrude your profile once out of the **Sketcher Work Bench**.

- 21.1 An unclosed profile as shown in Figure 1.34a. Notice the profile has a gap in it.
- 21.2 A profile with floating entities as shown in Figure 1.34b. Notice there is a line not attached to any other entity, it is floating.
- 21.3 Multiple profiles in one sketch as shown in Figure 1.34c. Notice both profiles are closed profiles but there are two of them. The two profiles have to be separate sketches.
- 21.4 An over constrained profile as shown in Figure 1.34d. Notice this example shows that one line is being dimensioned two different ways.

Figure 1.34 Profiles that can not be extruded



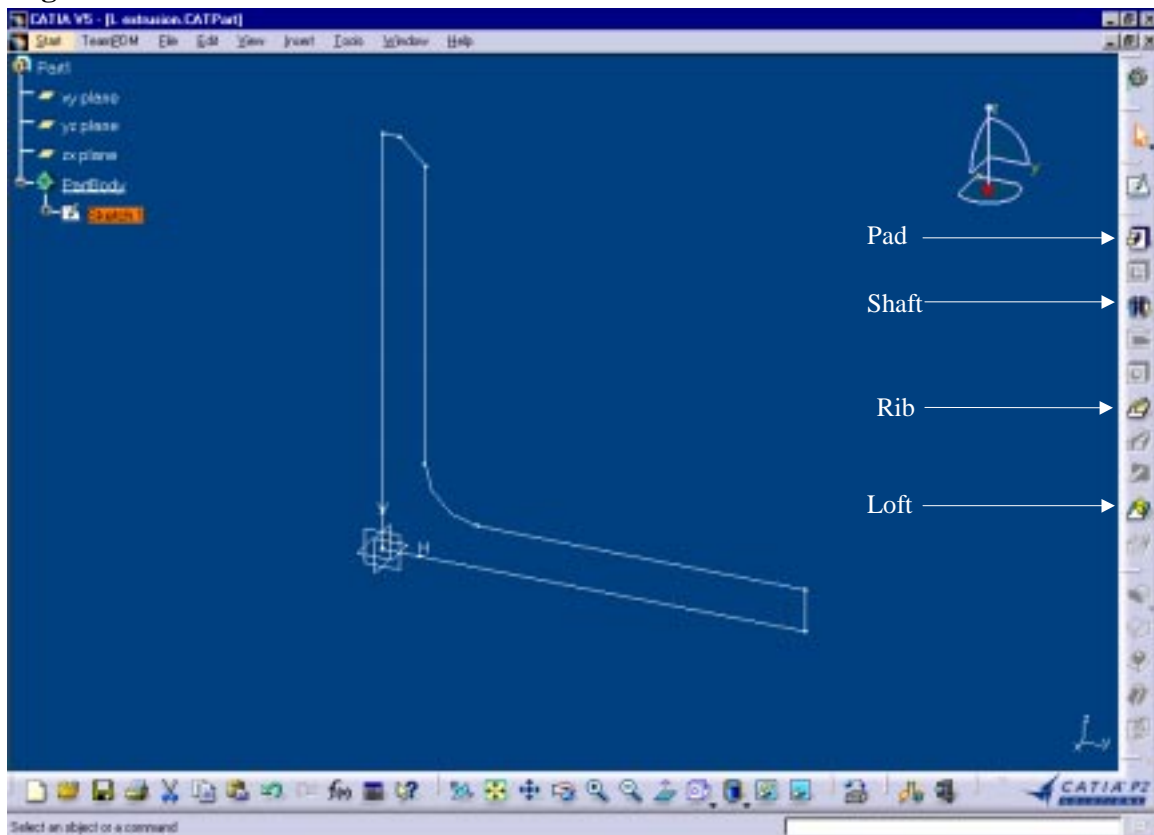
You can exit the **Sketcher Work Bench** with your profile in any of the above conditions, but CATIA V5 will not extrude the profile into a 3 dimensional part.

If you are ready to exit the **Sketcher Work Bench** select the **Exit** icon . The **Exit** icon is located in the top right of the **Sketcher Work Bench**.

Notice the profile rotates back to the original three dimensional view with your newly created profile of the **“L Shaped Extrusion”**. The **Sketcher Work Bench** grid disappears. The tools on the right hand tool bar will change, as shown in Figure 1.35. The only tools available for your use at this time are **Pad**, **Shaft**, **Rib** and **Loft**. The **Pad** tool is covered in Step 22 and Lesson 2. The **Shaft**, **Rib** and **Loft** tools are covered in the Advanced CATIA V5 Workbook. The next step will tell you how to use the **PAD** tool.

If your screen looks similar to Figure 1.35, you are now in the **Part Design Work Bench** and ready to go to Step 22.

Figure 1.35



22 Extruding The Newly Created Profile Using The Pad Tool



This step will put your newly created profile of the “**L Shaped Extrusion**” to the test. This is where you find out if there are any problems with your profile sketch created in the **Sketcher Work Bench**.

If you haven’t selected anything in the work area since exiting the **Sketcher Work Bench**, your profile should still be highlighted. If it is not still highlighted, select the profile or select the Sketch branch from the **Specification Tree**. When the profile is highlighted you can select the **Pad** icon. This will bring the **Pad Definition** window up (Figure 1.37). As the Pad Definition window pops up you should notice your profile becomes 3 dimensional. The **Specification Tree** just added another branch, the **Pad Specification Branch**. At this point you can specify how long to extrude the profile. You can type it in or select the up arrow and watch the part grow. Select the down arrow and watch it shrink. You can reverse the direction and/or mirror the extruded length. If these are not enough options you can select the **More** button in the **Pad Definition** window (Figure 1.37). The **More** button will let you specify the start location **First Limit:** and the ending plane **Second Limit:** of the profile being extruded. The **More** button will allow you to select an extruded direction other than the default direction, which would be normal to the sketch plane.

Figure 1.36

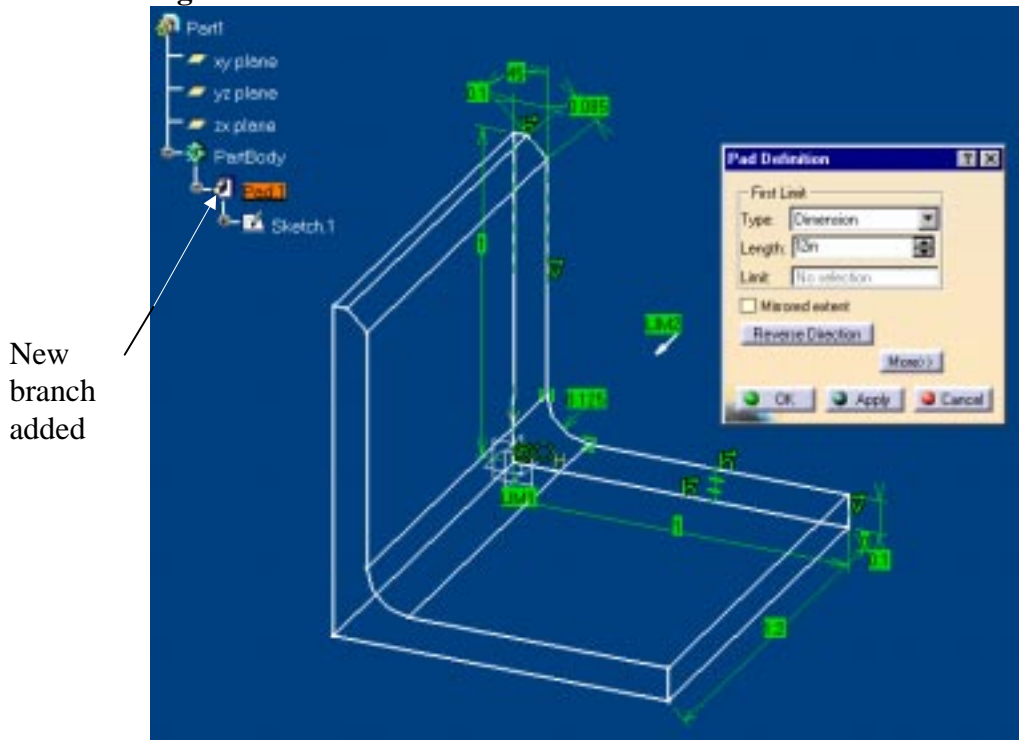
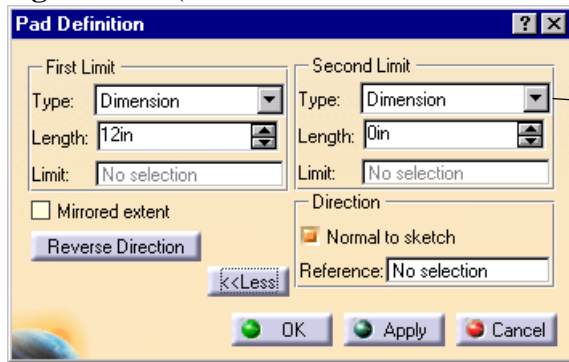
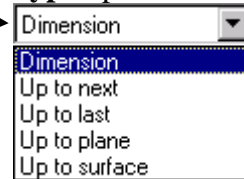


Figure 1.37 (Pad Definition window with More selected)



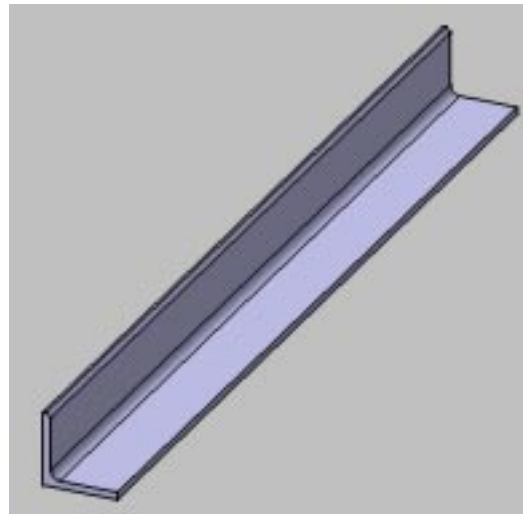
Type options available



These options are the same in the first and second

Once you have the **Pad Definition** window set up the way you want it, select the **Apply** button, this will give you a preview of what you just created. If you are not satisfied with the result select the **Cancel** button. If you are satisfied select the **Ok** button. The **Ok** button will create a three dimensional part from your sketch. For the “**L Shaped Extrusion**” extrude the profile 12 inches. Your extrusion should look like Figure 1.38.

Figure 1.38



23 Saving The Newly Created “L Shaped Extrusion”

You can stop what you are doing at any time and save the file you are working on. CATIA V5 allows the user to set the time period for the automatic save. Before saving and exiting make sure you have finished all operations you have started. If you save and/or exit in the middle of an operation, the operation will not be saved. CATIA V5 allows you to name the file as you wish. The file extension will be ***.CATPart**. All files created in the **Sketcher Work Bench** and **Part Design Work Benches** will have a ***.CATAPart** extension. To Save a CATIA V5 file complete the following steps:

- 23.1 Verify that all operations are complete and the part (CATPart) is the way you want it to be saved.
- 23.2 Select File from the top tool bar (Figure 1.39).
- 23.3 Select Save As (Figure 1.39).

- 23.4 In the **File, Save** window select the directory you want the CATPart saved in as shown in Figure 1.40.
- 23.5 In the same window type in the **File name**. For this lesson save the file as “**L Shaped Extrusion.CATPart**”. The extension is automatic.
- 23.6 Notice CATIA V5 will automatically give the file the extension “***.CATPart**”.
- 23.7 If everything is the way you want it in the **File, Save** window select the **Save** button.

Figure 1.39

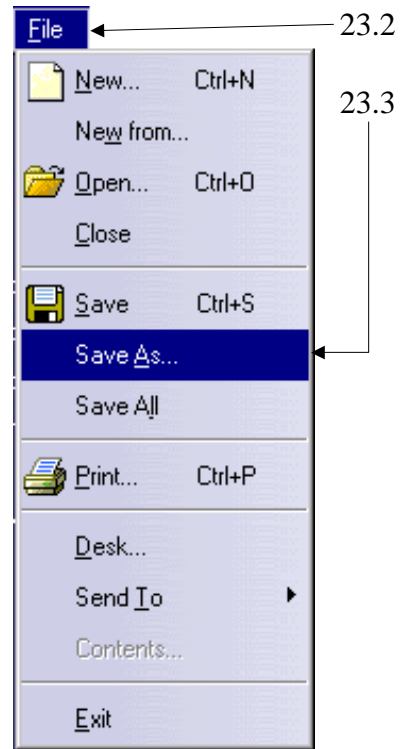
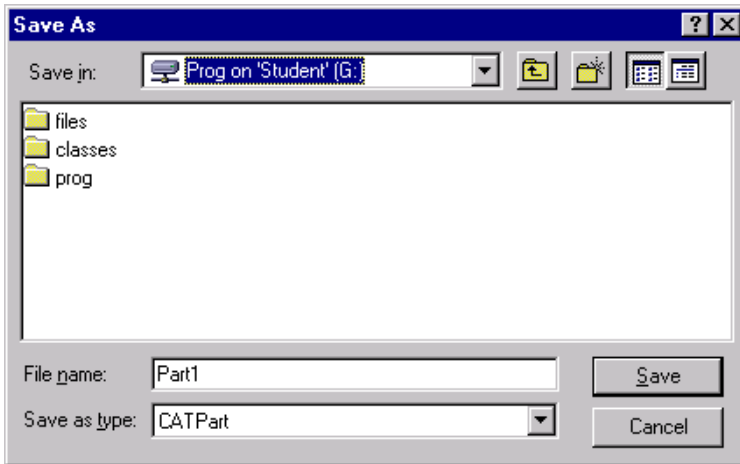


Figure 1.40



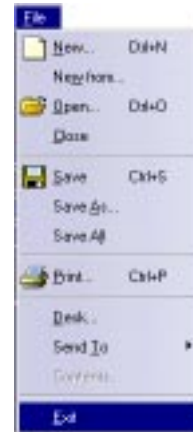
NOTE: Remember the file name and the directory you saved it to, you will need to reopen it to use in Lesson 2.

24 Exit CATIA V5

To exit CATIA V5 complete the following steps:

- 24.1 Make sure you saved the CATPart (if you wanted it saved). If you have made any unsaved changes to the CATPart and not saved, CATIA V5 will prompt you to save when exiting.
- 24.2 Select **F**ile from the top pull down tool bar as shown in Figure 1.41.
- 24.3 Select **E**xit.
- 24.4 If the CATPart was previously saved CATIA V5 will shut down and your computer will go back to the NT Desktop. As described above, if some changes were made to the CATPart without being saved, **CATIA V5 will prompt you to save before allowing you to exit to the NT Desktop.**

Figure 1.41



Lesson 1 Review

After completing this lesson you should be able to answer the questions and explain the concepts listed below:

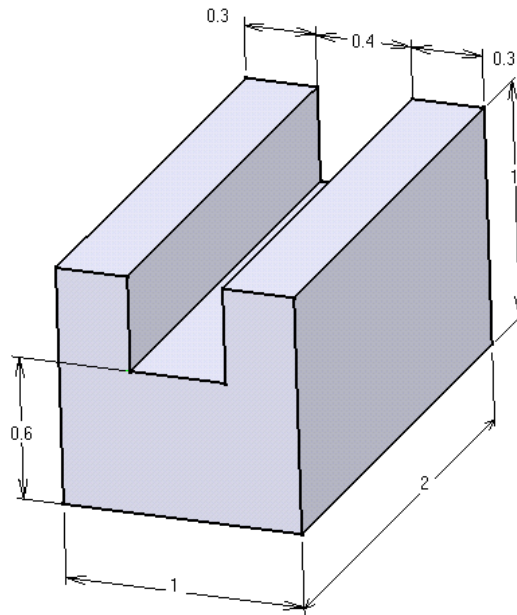
1. What is the definition of a constraint?
2. Does CATIA V5 require constraints to create a profile in the **Sketcher Work Bench**?
3. What is meant by an unclosed profile?
4. Can an unclosed profile be extruded?
5. What does anchoring the profile do in the **Sketcher Work Bench**?
6. How many different ways can you select the XY plane?
7. Explain how you would change the **Sketcher** units of measurements from mm to inches.
8. The **Sketcher** Grid is made up of two different entities, one is the **Primary Spacing**, name the other?
9. What is the advantage of constraining a profile in the **Sketcher Work Bench**?
10. How do you modify a constraint?
11. Is it a good thing to over constrain a profile?
12. Explain your answer to question 11.
13. What icon do you use to exit the **Sketcher Work Bench** and enter the **Part Design Work Bench**?
14. How can you view all the default tool bars in **Sketcher Work Bench**?
15. What tool in the **Part Design Work Bench** is used to extrude a profile created in the **Sketcher Work Bench**?
16. The actual process of extruding a profile adds what branch to the **Specification Tree**?
17. List as many types of constraints as you can.
18. Can one Sketch have more than one profile?
19. While in the **Sketcher Work Bench** and using the mouse how would you move (pan) the profile around the screen?
20. When you are connecting one end point of a line to another how does CATIA V5 let you know you are **Snapping** to the existing end point and not just getting close?

Practice Exercises:

Now that your CATIA V5 tool box has some tools in it, put them to use on the following practice exercises. The shapes are simple and can be completed in one sketch. The dimensions represent the constraints you are to use in the **Sketcher Work Bench**. The first practice exercise has the suggested steps to completing the task along with some helpful hints. Each subsequent practice exercise contains less suggested steps and helpful hints. By the last practice exercise you will be on your own!

Each practice exercise has a suggested name to use when saving the exercise. It is critical that you use the suggested name so you can find the correct CATPart if it is used in a later lesson. Good Luck!

- 1.) Using the **Sketcher Work Bench** and the other tools covered in Lesson 1 create the following profile and extrude to the dimensions shown below. When completed save as “**Lesson 1 Exercise 1.CATPart**”.

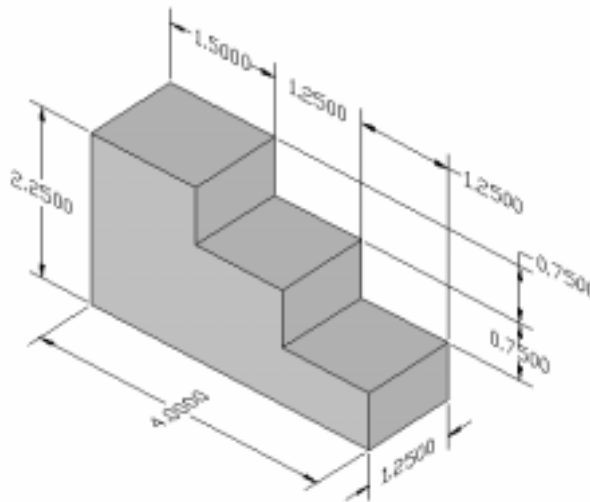


Suggested Steps:

1. Select the XY plane (the plane the profile will be sketched on). Reference Step 3 for information on selecting planes.
2. Enter the **Sketcher Work Bench**. Reference Step 4.
3. Sketch the profile of the part.
Hint: use the **Profile** tool.
4. Anchor the lower left hand corner of the sketch. Reference Step 17 for anchoring a profile.

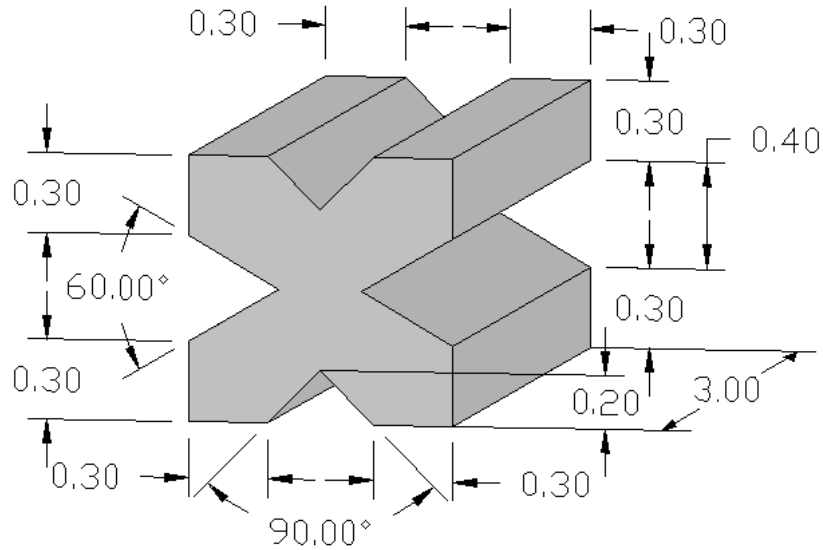
5. Constrain the profile to match the dimensions shown above. Reference Step 18 for constraining a profile.
6. Exit the **Sketcher Work Bench**, return to the **Part Design Work Bench** (the 3D environment). Reference Step 21 for exiting the **Sketcher Work Bench** and entering the **Part Design Work Bench**.
7. Once in the **Part Design Work Bench** extrude the profile to the dimension shown (2"). Reference Step 22 for extruding a profile.
8. Save the part as "**Lesson 1 Exercise 1.CATPart**". Reference Step 23 for saving a file.

- 2.) This part (profile) should be straightforward. This would be a good exercise to try different methods of constraining and testing the results. Save the shape as "**Lesson 1 Exercise 2.CATPart**".



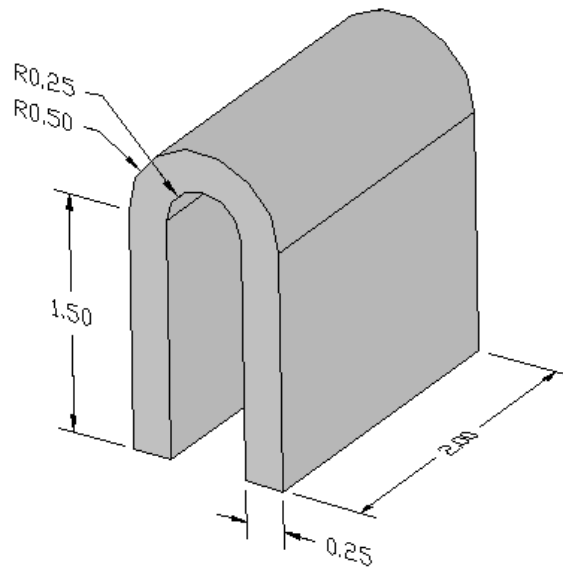
- Hint:** To help make it easier to sketch this part set the grid **Primary Spacing** to 1 and the **Graduations** to 4. This will put the grid lines in the **Sketcher** screen to a .25 inch spacing. With that spacing all you have to do is snap to the intersections of the grid to sketch the part.

- 3.) This practice exercise is a little bit more challenging, lets see what you can do with it. Save this CATPart as "**Lesson 1 Exercise 3.CATPart**".




Hint: It is not as complicated as it looks. If your grid **Graduations** are set to 10 just snap to the intersections for the beginning and ending points of your lines. To set the constraint for the angles select the angled lines and the angle constraint will appear. Reference Step 19 for modifying the angle value. If the profile gets over constrained delete the **Parallel** constraint. Save the file as “**Lesson 1 Exercise 3.CATPart**”.

4.) This practice exercise should challenge you. For this part use radius values, not angles. Save this CATPart as “**Lesson 1 Exercise 4.CATPart**”.

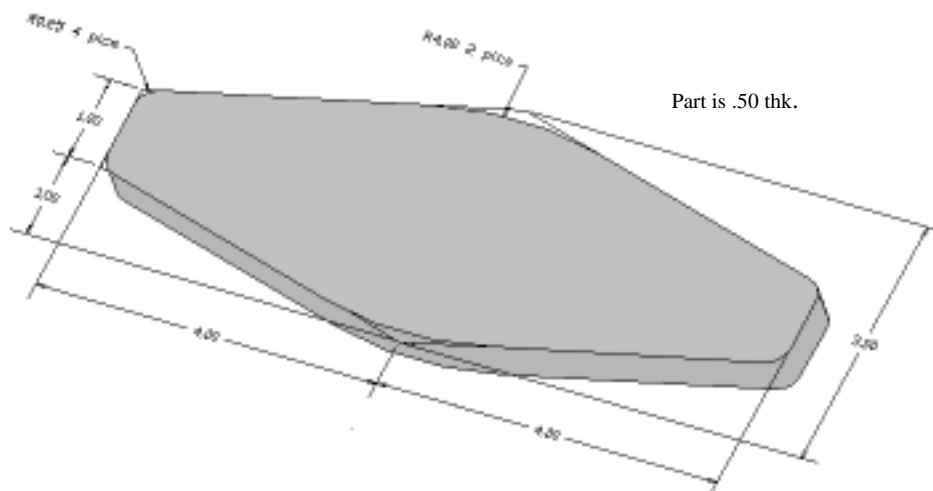


Hint: This part can be done using the radius option in the profile command.
 Before starting, set the grid **Primary Spacing** to 1 and the **Graduations** to 4.

Sketching with the Profile icon (radius option)

1. Starting at the bottom left corner of the part.
2. Select the **Profile** icon from the right menu bar. 
3. Sketch the vertical 1.50 inch line that defines the left edge of the part.
4. Now sketch the first arc along the top of the part. To do this hold down the left mouse button and drag it in the direction you want the arc to go then release the mouse button. The arc will appear and allow you to drag and place it where you want. Place it on the grid intersection 2 inches above the bottom of the part and a half-inch to the right. This will only create half of the arc needed, so the process will have to be repeated to sketch the other half of the arc.
5. Finish sketching the rest of the part. When you reach the inside .25 radius, just repeat Step 4.
6. When the sketch is done constrain it to double check that all the dimensions match the part shown above. Make the necessary changes if needed.

5.) This will give you more practice using the line and corner icons. Save this CATPart as **“Lesson 1 Exercise 5.CATPart”**.



Hint: Use the **Line** or **Profile** icon first to sketch the profile using sharp corners (no radius). Once it is constrained to the dimensions above, go back and add in the radiuses using the **Corner** icon.