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Introduction

CATIA Version 5 Part Design and Sketcher

Upon completion of this course the student should have a full understanding of the following topics:

- Creating sketches
- Constraining sketches
- Modifying sketches
- Creating parts
- Modifying parts
- Performing boolean operations on parts
- Basic use of surfaces in part design
- Applying materials to parts

Manual Format

It is important to understand the format of the manual in order to use it most effectively. This manual is designed to be used along with an instructor; however, you will need to do a lot of reading as well, in order to fully understand CATIA Version 5. The exercises in this book will list steps for you to complete, along with explanations that try to inform you what you have just done and what you are getting ready to do. The actual steps are in bold type and the information that follows the steps is for your benefit. Anything that appears in *italics* refers to a message CATIA provides—this includes information in pull-down menus, pop-up windows and other messages.

An example of a step and its explanation is shown below (note: normally the lines will not be there):

Select a location to the right of the origin. This specifies the other end point of the line. You will continue specifying locations in order to complete your profile. It should appear similar to the diagram shown below.

As you can see, the desired action blends in with the text except that it appears in bold. The information following the step explains what that step accomplished and where you are going next. It is important to read this information in order to better your understanding of CATIA Version 5.

Also, you will find that the exercises build upon themselves. Later exercises often assume you know how to do certain steps which have been covered earlier in the course. If you did not quite pick up what you needed to know from an exercise, you will probably want to review it several times before moving onto more advanced sections. The advanced sections assume that you have a good understanding of the previous sections therefore fewer steps will be provided. Eventually, you are expected to be able to create parts without any steps.

Part Design & Sketcher

CATIA Version 5 uses the Sketcher workbench as its principal method to create profiles. These profiles can be constrained using many different types of constraints. The first objective of the course is to learn to use the Sketcher and constrain your profiles to the desired specifications. If you have used the Dynamic Sketcher from CATIA Version 4, this will look very similar. Otherwise it is a new environment and it can be frustrating at first, especially if you already know CATIA Version 4. However, in time you will find that it is a very powerful method for creating profiles, and is easy to use.

The second objective of the course is to use these sketches in part design. The sketches are used to define the two-dimensional cross-sections to be used to design three-dimensional shapes. There are a few different shapes that can be made as well as various operations that can be performed on them. By combining these shapes and operations, you can design a variety of parts.

The third objective of the course is to familiarize you with the advanced methods of creating sketches and parts. This includes using construction geometry and projecting three-dimensional geometry to the sketch plane. This will include the use of formulas to set up typical values at multiple locations as well as more complex formulas to provide a more dynamic sketch. In terms of part design, you will learn how to use multiple parts and perform boolean operations on them.

The fourth objective is to become efficient at modifying your designs. You can modify your design either by changing the parameters of a part operation or by modifying the sketch that was used. In CATIA Version 5 this is fairly simple, and the modification of your design is the real strength of part design.

The fifth objective is to introduce the use of wireframe and surfaces in the part design process as well as applying various materials to your design. This is meant only to be an introduction and not a complete course on these subjects.

In conclusion: You should be able to design many parts using the Sketcher and the Part Design workbenches of CATIA in an efficient manner. As mentioned before, you may find it frustrating at first but very natural by the end of the course.

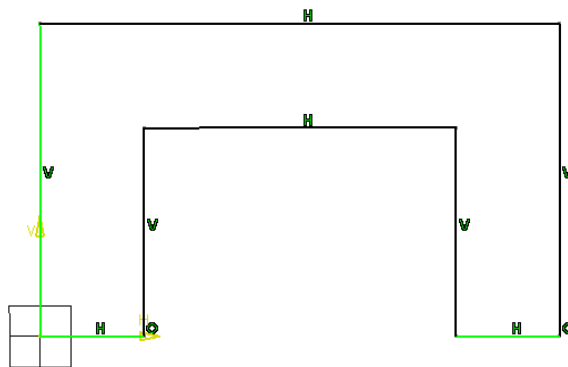
Profiles

This section will discuss the profile icon which is the most commonly used icon when defining sketches. The purpose of these exercises is to demonstrate the usefulness of the profile icon and how to use it effectively. You can use it to generate basic shapes or more involved shapes all in one operation.

The most common use is to specify corner points of your desired profile and it will generate lines between those points until you either select on the profile icon again, double select a location or select a location that closes the profile. You can use one of the other two sub-options that are located next to the sketch tools to generate curves as you are defining the profile. You can also generate a tangent curve using the mouse while sketching with the profile icon. This will be done in a later exercise.

As you are defining the profile, occasionally the element will appear in blue before defining the endpoint or a constraint may appear in blue. If you select the endpoint while the element or the constraint is blue then it will automatically put those constraints on the geometry when it is created. This is useful when defining horizontal and vertical lines because if they appear in blue while you are defining them then the horizontal and vertical constraint will automatically be generated on the element.

You will now build various profiles to get experience using the various capabilities of the profile icon. The first profile you are going to build looks like the one shown below.

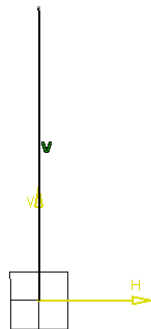


Start a new part and go into the Sketcher workbench with the yz plane.

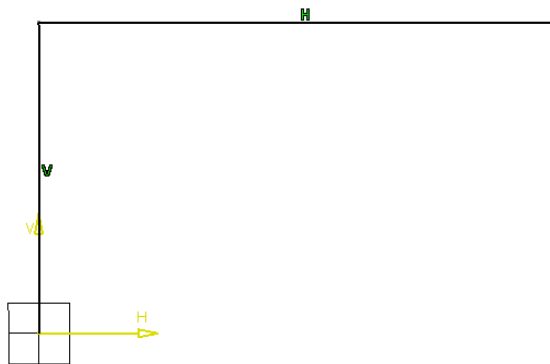
Select the Profile icon.  It should be highlighted.

Select the origin point of the sketch plane. This specifies the starting point for your profile. When you specify the next location, make sure the line appears blue before selecting the location. This will put the vertical constraint on your element automatically.

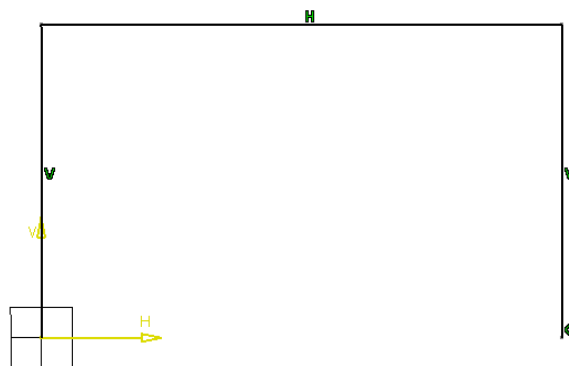
Select a location above the origin. If the line appeared blue before you selected the second location then it should appear with the vertical constraint on it like the one shown below. Make sure when you specify the other locations that those elements appear blue as well before selecting the location.



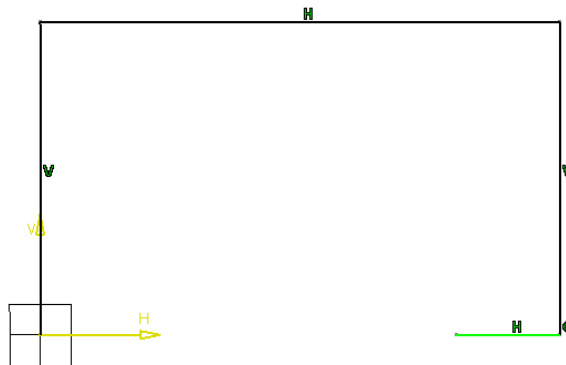
Select a location to the right of the previous location. It should appear with the horizontal constraint on the element and look similar to the diagram shown below.



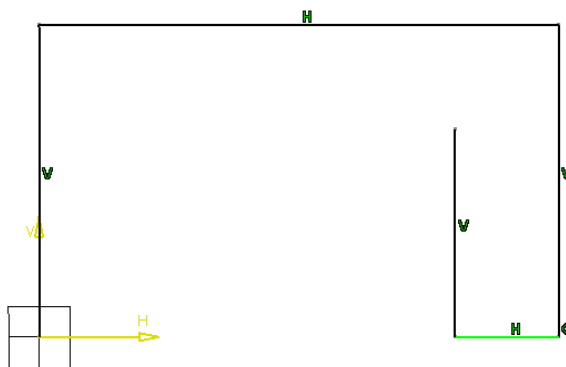
Select a location below the previous location along the H axis. It should appear similar to the diagram shown below. You might notice a little green circle appear, this is a coincidence constraint. This coincidence constraint forces the end point to be aligned with the H axis. This and other constraints will be discussed in more detail in later exercises.



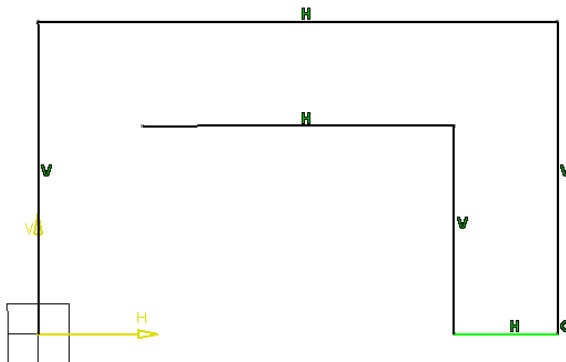
Select a location to the left of the previous location. It should appear similar to the diagram shown below. You might notice that the little horizontal line appears green. This means that the element is iso-constrained. This will be discussed in more detail in later exercises.



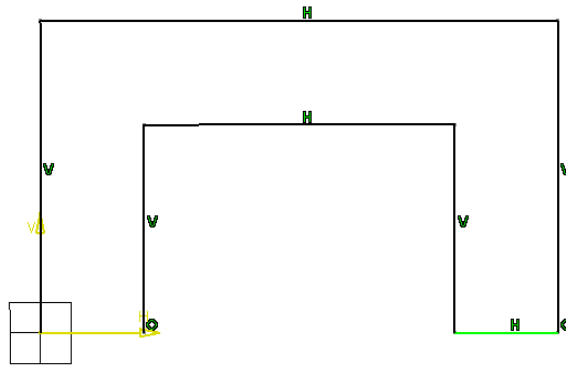
Select a location above the previous location. It should appear similar to the diagram shown below.



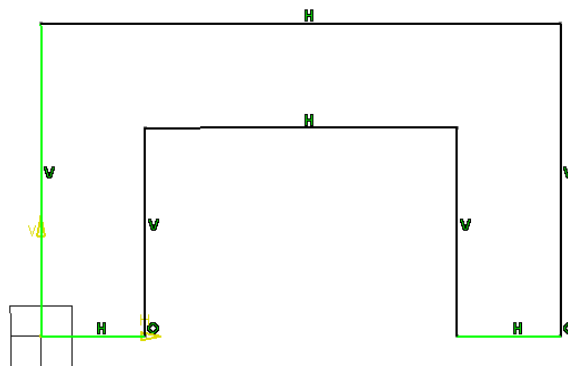
Select a location to the left of the previous location. It should appear similar to the diagram shown below.



Select a location below the previous location. It should appear similar to the diagram shown below.



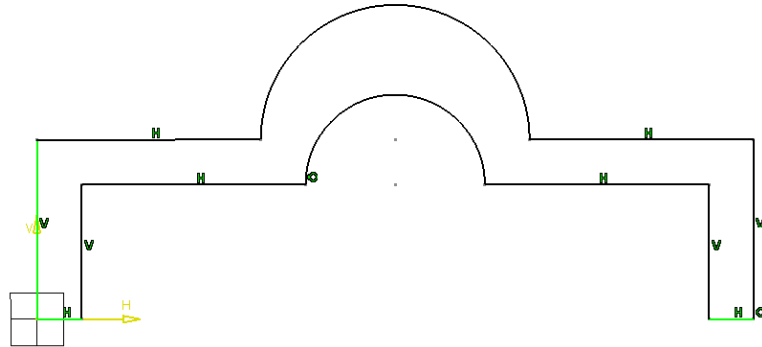
Select the origin point of the sketch again. The profile should appear similar to the diagram shown below and exit the profile icon.



If you desire you can save your document and call it profile1 with your initials.

Save and close your document.

You will now perform a sketch of a part using the three point arc icon which is a sub-option of the profile icon. It will appear to the right of your sketch tools. This icon allows you to define a three point arc while using the profile icon. The second profile you are going to build looks like the one shown below.

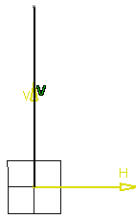


Start a new part and go into the Sketcher workbench with the yz plane.

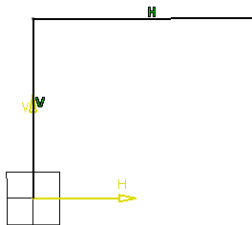
Select the Profile icon.  It should be highlighted.


Select the origin point of the sketch plane. This specifies the starting point for your profile. When you specify the next location make sure the line appears blue before selecting the location. This will put the vertical constraint on your element automatically.

Select a location above the origin. If the line appeared blue before you selected the second location then it should appear with the vertical constraint on it like the one shown below. Make sure when you specify the other locations for the lines that they appear blue as well before selecting the location.



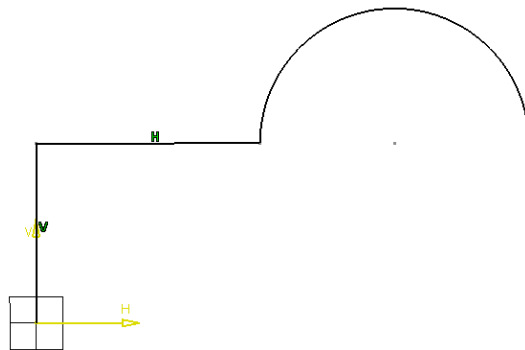
Select a location to the right of the previous location. It should appear similar to the diagram shown below.



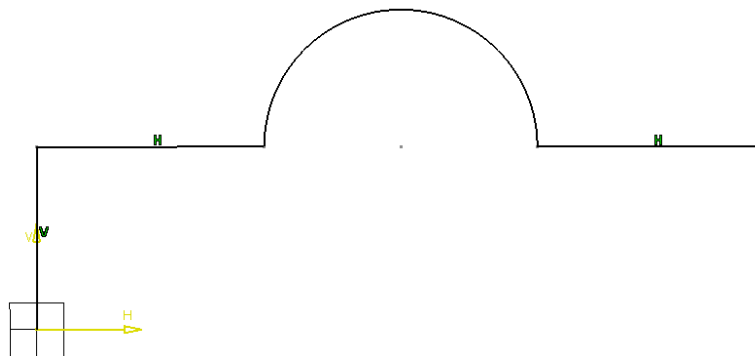
Select the Three Point Arc icon.  It is located to the right of the sketch tools. This icon will allow you to specify a location for the arc to pass through and an ending location for the arc. The arc will begin at the last location specified which is the endpoint of the line.

Select up and to the right of the previous location. This specifies the location that the arc should pass through. The next point specifies the endpoint of the arc.

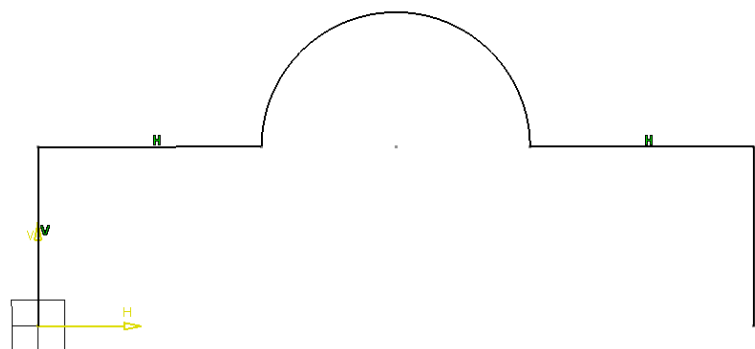
Select down and to the right of the previous location. This location should be straight across from the start of the arc. It should appear similar to the diagram shown below. Notice how the three point arc icon next to the sketch tools automatically turned off and the line icon turned on.



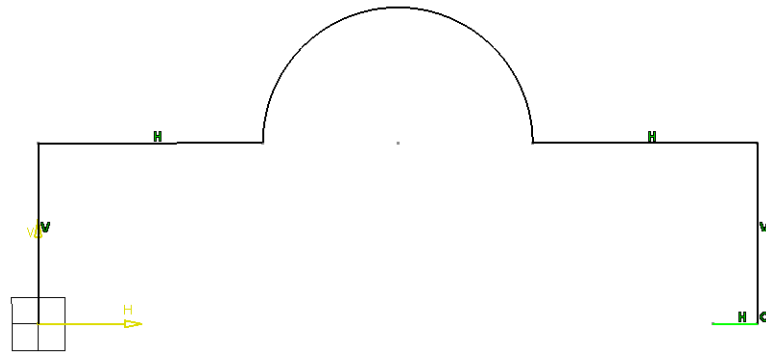
Select to the right of the previous location. It should appear similar to the diagram shown below.



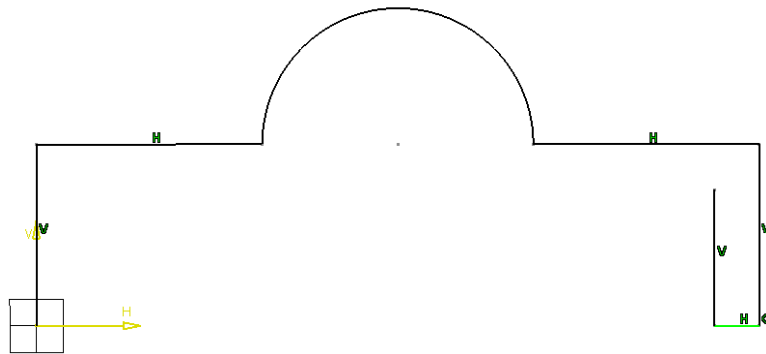
Select below the previous location. It should appear similar to the diagram shown below.



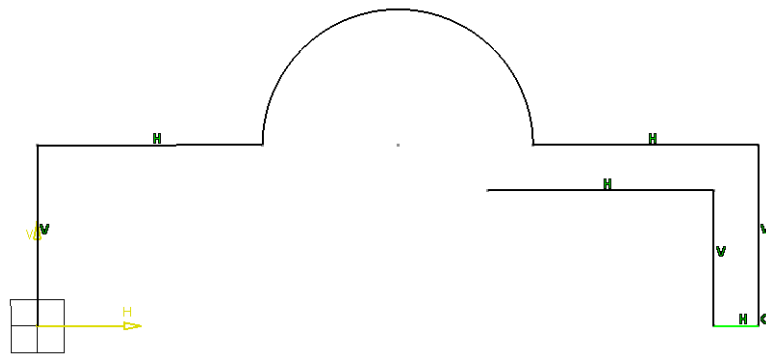
Select to the left of the previous location. It should appear similar to the diagram shown below.




Select above the previous location. It should appear similar to the diagram shown below.



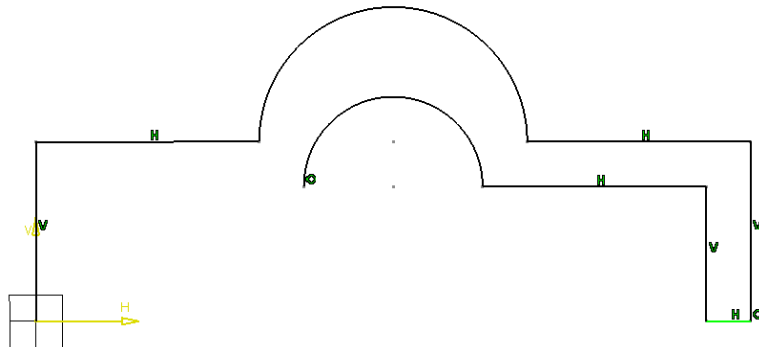
Select to the left of the previous location. It should appear similar to the diagram shown below.



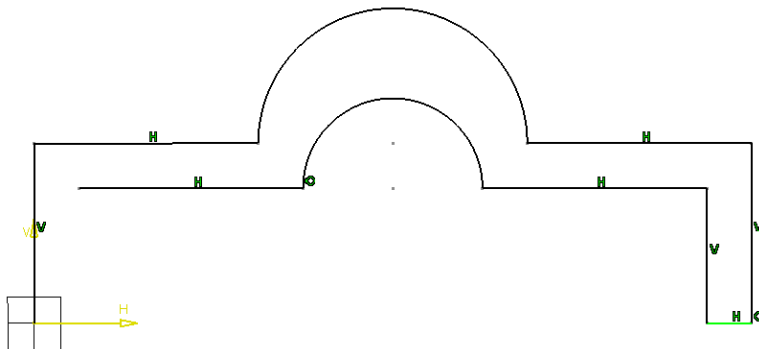
Select the Three Point Arc icon.  It is located to the right of the sketch tools. This icon will allow you to specify a location for the arc to pass through and an ending location for the arc. The arc will begin at the last location specified which is the endpoint of the line.

Select up and to the left of the previous location. This specifies the location that the arc should pass through. The next point specifies the endpoint of the arc.

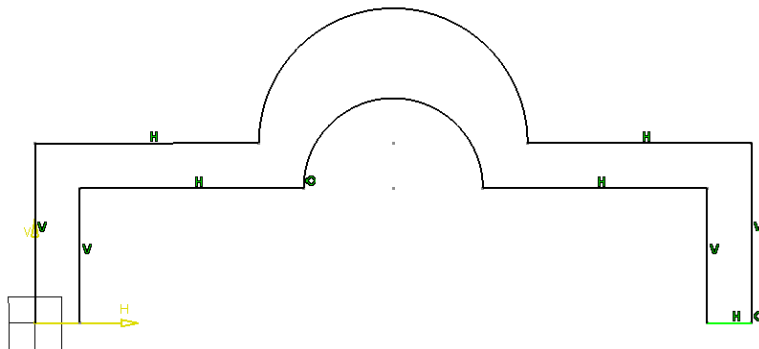
Select down and to the left of the previous location. This location should be straight across from the start of the arc. It should appear similar to the diagram shown below. Notice how the three point arc icon next to the sketch tools automatically turned off and the line icon turned on.



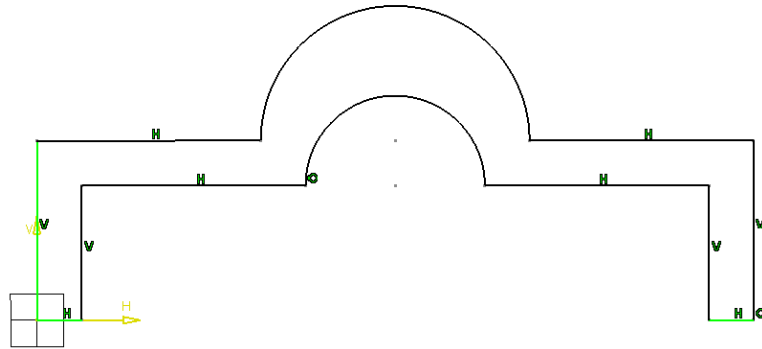
Select to the left of the previous location. It should appear similar to the diagram shown below.



Select below the previous location. It should appear similar to the diagram shown below.



Select the origin point of the sketch again. The profile should appear similar to the diagram shown below and exit the profile icon.

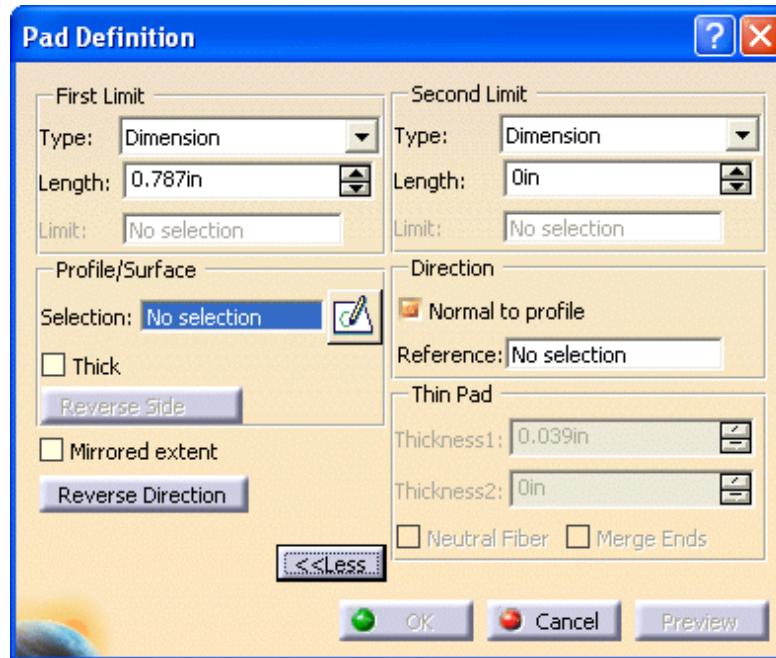


If you desire you can save your document and call it profile2 with your initials.

Save and close your document.

Pad

The pad icon allows you to use a sketch and extrude it in a linear direction producing a solid pad. You can create a sketch or profile on-the-fly by pressing the third mouse button while in the *Selection* box. This would allow you to use one of the available options to define the profile if you did not have it created already. When you create a pad, a *Pad Definition* window appears like the one shown below.



Initially the window will appear with only the *First Limit* and then you have the option to select the *More>>>* option to see the *Second Limit*. Since the options are the same for both limits they will be discussed only once.

Type


<i>Dimension</i>	Allows you to key in a <i>Length</i>
<i>Up to next</i>	Goes to the next side of an existing part
<i>Up to last</i>	Goes to the last side of an existing part
<i>Up to plane</i>	Goes to a specified plane which is its <i>Limit</i>
<i>Up to surface</i>	Goes to a specified surface which is its <i>Limit</i>

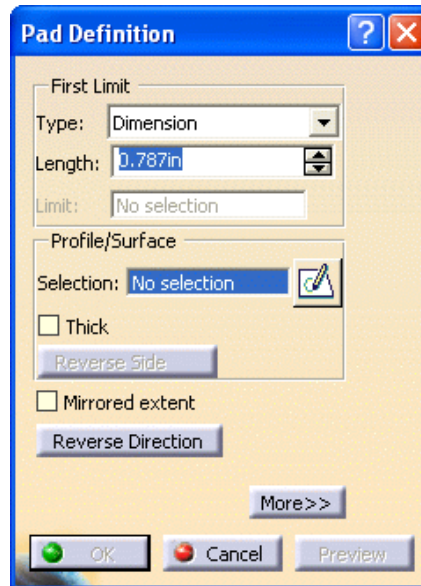
When you select a *Type* other than *Dimension* you will have the option to specify an *Offset* value from the corresponding limit.

Profile/Surface

<i>Selection</i>	Specifies which sketch will be used, you have the option to modify the sketch using the sketcher icon next to the box. You can select a surface instead and use the surface as your profile.
<i>Thick</i>	Toggles the <i>Thin Pad</i> option. This allows you to add thickness to the elements that make up the sketch.
<i>Reverse Side</i>	Reverses the side an open profile will use to determine its shape
<i>Mirrored extent</i>	Applies to the <i>Type Dimension</i> , it will go the same distance in both directions, thereby not being able to specify a second limit
<i>Reverse Direction</i>	Changes the direction to the opposite direction
<i>Direction</i>	
<i>Normal to profile</i>	The direction will be in the normal direction of the sketch
<i>Reference</i>	Allows you to specify an element that defines the direction
<i>Thin Pad</i>	
<i>Thickness1/2</i>	Specifies the thickness that will be applied to each sketch element
<i>Neutral Fiber</i>	Forces the sketch element to be in the center and the thickness is added to both sides equally
<i>Merge Ends</i>	Extends or trims the elements to existing material

Open the Pad1 document and save with your initials. You should see two sketches already created for you.

Select the Pad icon.  This will allow you to create a pad using one of the sketches. This exercise is going to cover the various methods that you can use to create pads. A *Pad Definition* window should appear similar to the one shown below.



Select *Sketch.1*. This specifies that you want to use that sketch to define the profile of your pad. For this pad you are going to use the basic option of keying in a length. You will also preview what the *Mirrored extent* and *Reverse Direction* options allow you to do.

Change the value in *Length* to be 4. Do not press Enter or else it will automatically create the pad with that value. Normally you would just enter the value and press Enter, however you are going to want to *Preview* in order for you see what it is going to do until you understand the different options.

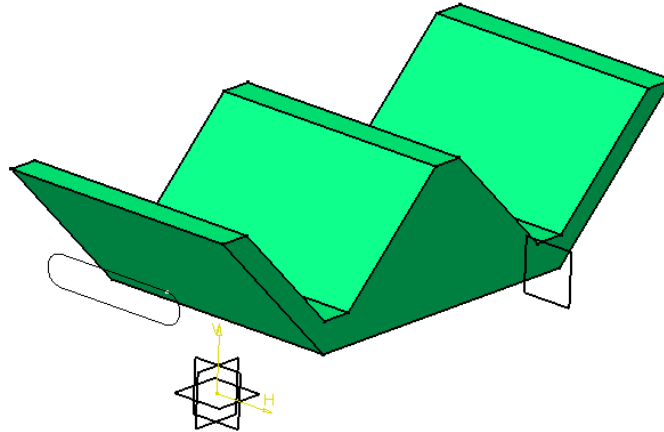
Select *Preview*. A preview of what the pad will look like appears. You will now change some of the other options to see the difference between them.

Select *Mirrored extent* and select *Preview*. As you can see, instead of the pad extending in only the one direction, it now extends both directions, four inches each. It basically is using your current sketch as the mirror plane.

Select *Mirrored extent* again to turn it off and select *Preview*. Now you are going to reverse the direction in order for the pad to be created in the opposite direction.

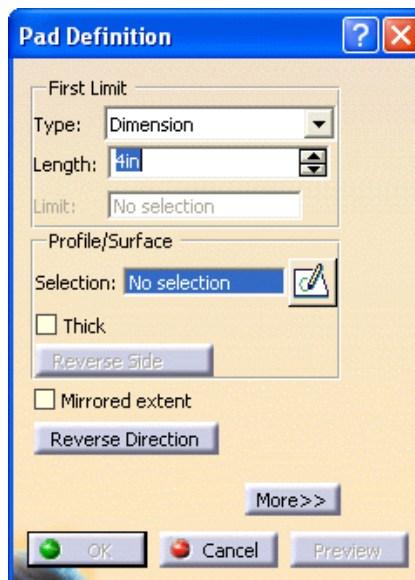
Select *Reverse Direction* and select *Preview*. Notice that the pad is still going to be four inches wide but it is now going in the opposite direction. This is the pad you want to create.

Select OK. The pad should be created and appear similar to the diagram shown below. Notice that the sketch automatically was hidden after being used by the pad. This is true when using most of the options because of a setting under the pull down menu *Tools, Options*.



You are now going to explore some of the other *Types* that you can use to define limits for pads that you create.

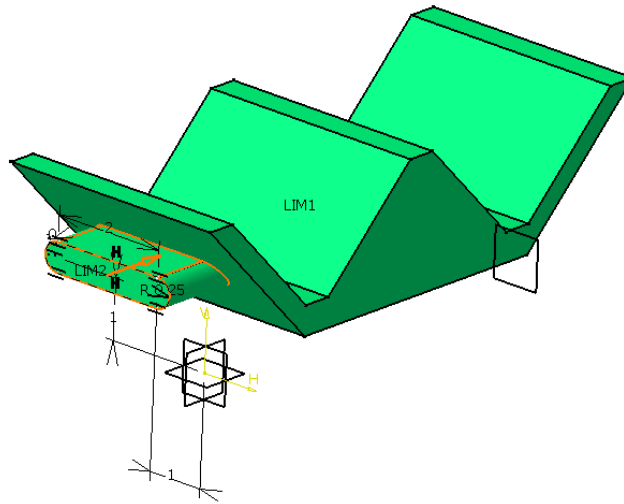
Select the Pad icon.  A *Pad Definition* window appears as shown below.



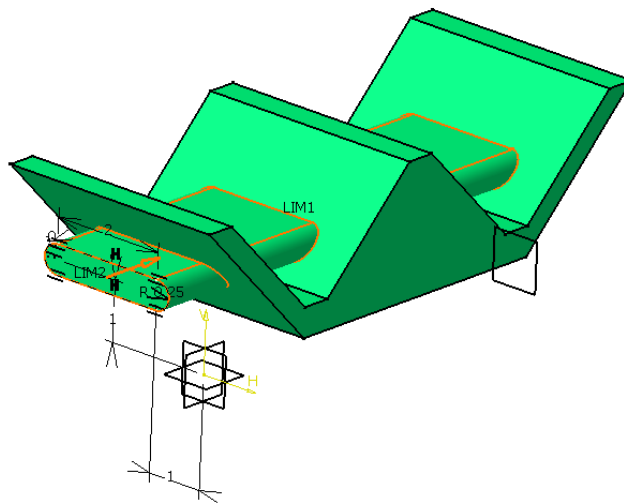
Select Sketch.2. This specifies the sketch that you want to use to create the next pad.

Select Reverse Direction so that the direction is toward the other pad. Now you are going to see what the other *Types* allow you to do.

Change the *Type* to *Up to next* and select *Preview*. Notice that the pad only goes to the next side of the other pad. It should appear similar to the diagram shown below.

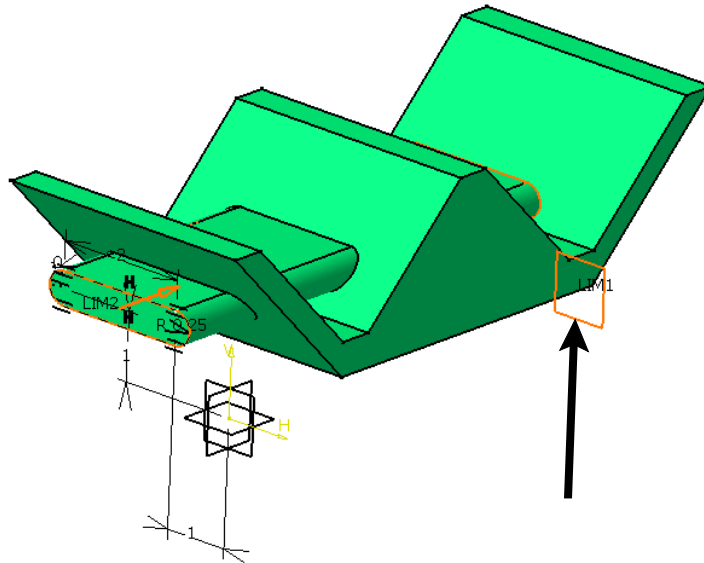


Change the *Type* to *Up to last* and select *Preview*. Notice that the pad goes all the way to the last side of the previous part. It should appear similar to the diagram shown below.



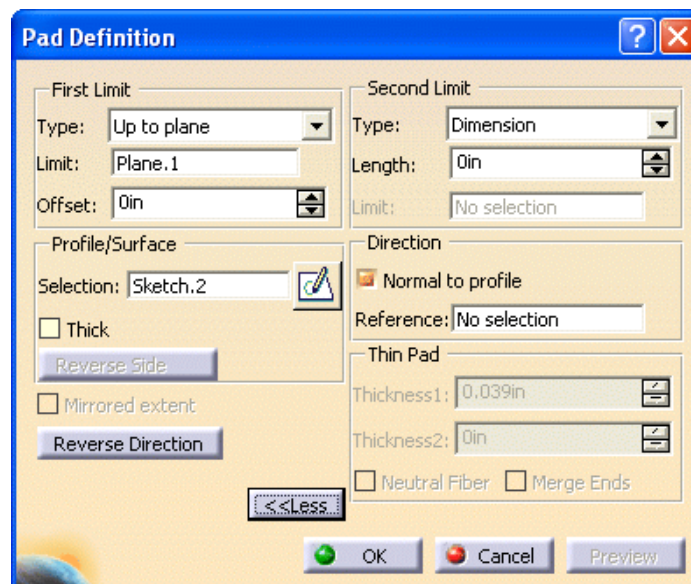
Change the *Type* to *Up to plane*. When you use this option you have to specify a plane or a planar side that you want the pad to be limited by.

Select the plane that is away from the origin as shown below and select *Preview*.
Notice that the pad goes up to the plane and then stops. It should appear similar to the diagram shown below.



You may have to rotate the part around in order to see the limitation better. The *Up to surface* option works very similar to the *Up to plane* option except that you can specify a surface instead of a plane.

Select the *More>>* option. This expands the window and shows some other options. The window should appear similar to the one shown below.

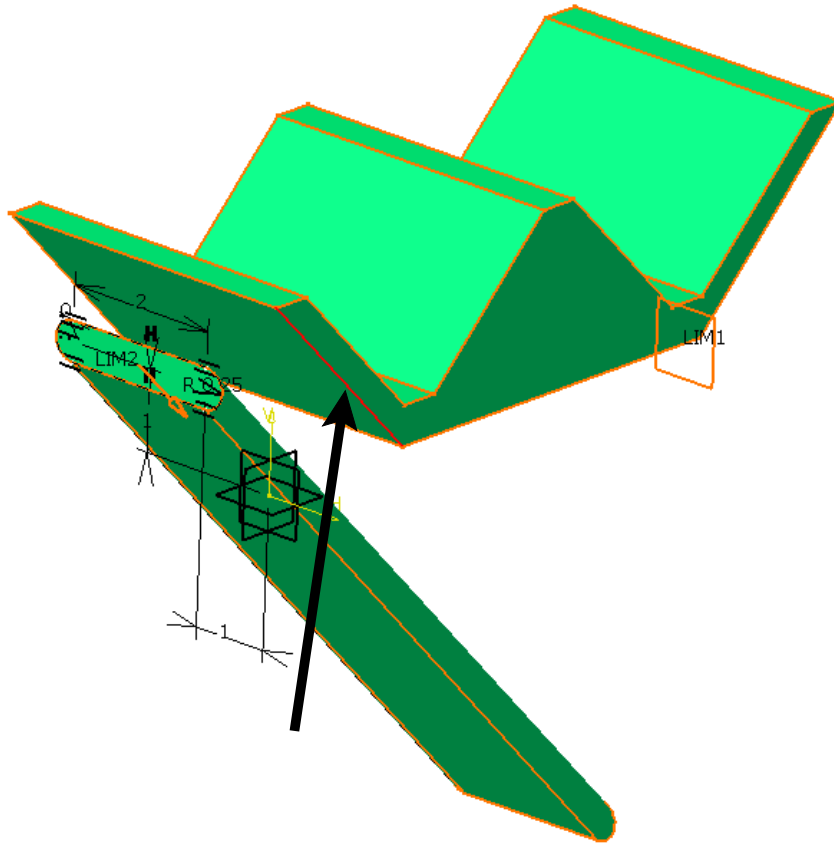


Currently the *Direction* is specified to be *Normal to profile*. You will turn that off and specify an element to be used as the direction. Once again this is just to show you the capabilities of the option.

Select *Normal to profile* to turn it off. The *Normal to profile* option is no longer activated.

Select in the *Reference* box. This allows you to specify an element to be used as the direction.

Select the angled edge closest to the origin as shown below and select *Preview*. The pad extrudes in the direction of the line and stops at the plane that was specified earlier. It should appear similar to the diagram shown below.



Select *Normal to profile*. This changes the direction back to being normal to the sketch. You are now going to use a *First Limit* and a *Second Limit* to create the pad.

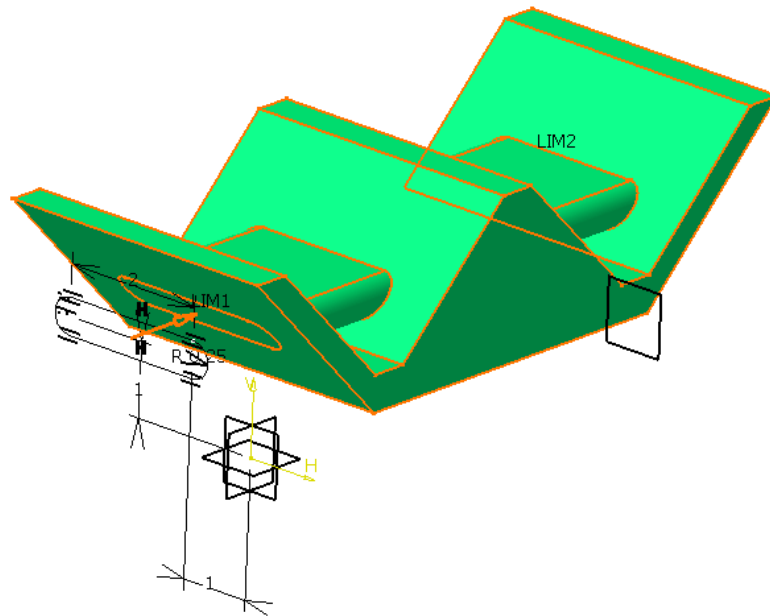
Under the *First Limit* select the *Limit* box. This will allow you to specify a new plane for your limit.

Select the angled side closest to the sketch. This defines the *First Limit*. You will now define the *Second Limit*.

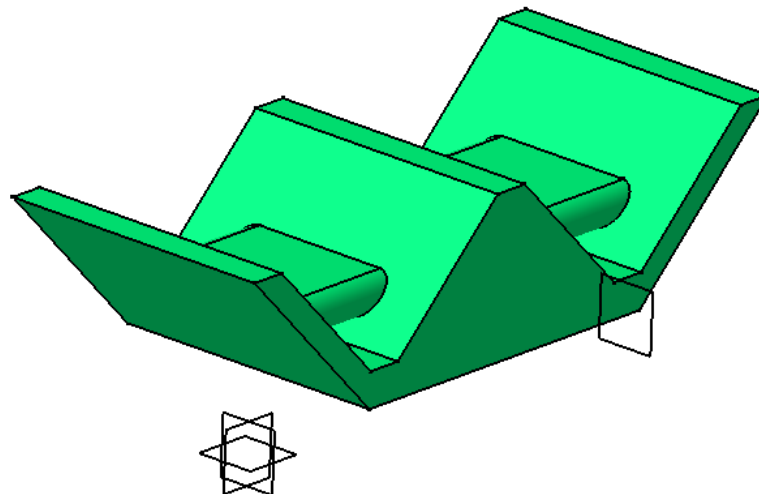
Under the *Second Limit* change the *Type* to *Up to plane*.

Under the *Second Limit* select the *Limit* box.

Select the angled side farthest from the sketch and select *Preview*. This defines the *Second Limit* and shows you a preview of your new pad. It should appear similar to the diagram shown below.



Select *OK*. The final part should look similar to the diagram shown below.




This exercise showed most of the options available when creating a pad. There are other shapes that have these same options and they work the same. Hopefully you have a good understanding of what each option allows you to do.

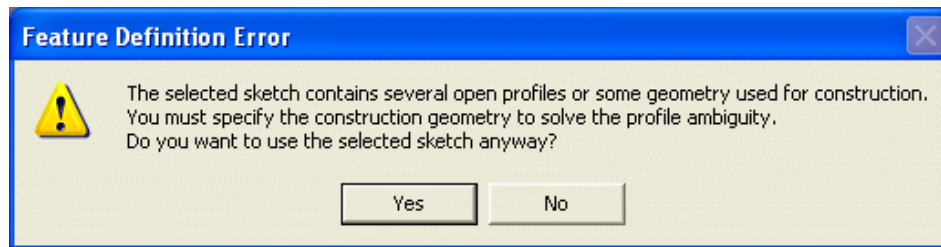
Note: Open profiles (sketches) can be used to create pads or pockets as long as they will be closed by the other faces of your existing part.

Save and close your document.

Open the Pad2 document and save with your initials. You should see a sketch already created for you. You are going to use the *Thin Pad* options to finish the model.

Select the Pad icon.  This will allow you to create a pad using the sketch. The *Pad Definition* window appears.

Select *Sketch.1*. This specifies that you want to use that sketch to define the profile of your pad. A *Feature Definition Error* window appears. This error message appears since your sketch does not contain closed profiles. However, this is okay since you are going to use the *Thin Pad* options.



Select *Yes*.

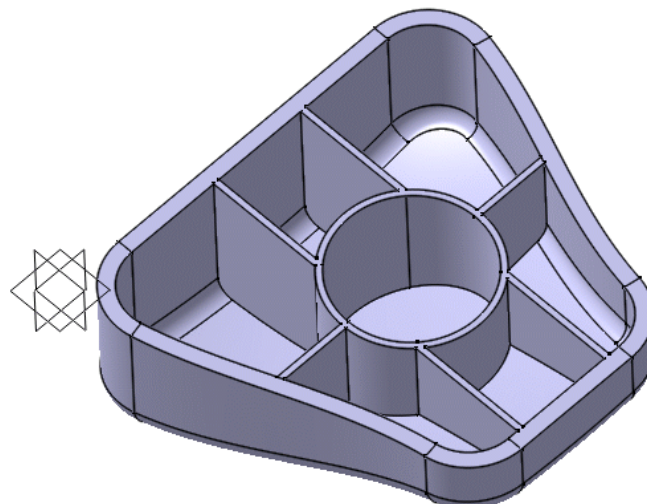
Turn the *Thick* option on. The *Thin Pad* options become available.

Turn on the *Neutral Fiber* option and specify **0.1 for *Thickness1*.**

Make sure the direction is pointing downward. If the direction is pointing upward, then select the *Reverse Direction* button.


Change the *First Limit* to be *Up to surface* and select the outer surface of the part. You will have to rotate the part in order to select the outside surface of the part.

Select *OK*. The pad is created and the part should appear similar to the diagram shown below.



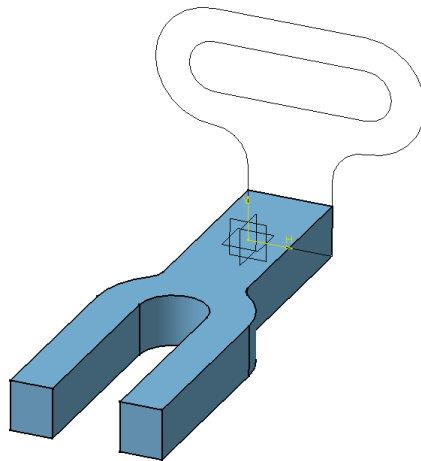
Save and close your document.


Open the Pad3 document and save with your initials. You should see three sketches already created for you.

Select the Pad icon.  This will allow you to create a pad using one of the sketches.

Select *Sketch.1*. This specifies that you want to use that sketch to define the profile of your pad.

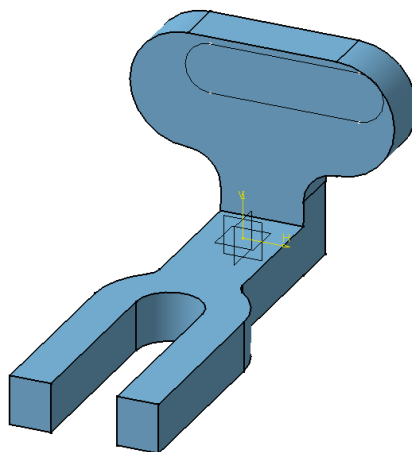
Using the *Type Dimension* and a *Length* of 0.75 create the pad by selecting *OK*. The pad should appear similar to the diagram shown below.



Select the Pad icon.  This will allow you to create a pad using one of the sketches.

Select *Sketch.2*. This specifies that you want to use that sketch to define the profile of your pad.

Using the *Type Dimension* and a *Length* of 0.75 create the pad by selecting *OK*. The pad should appear similar to the diagram shown below.



You will continue with this exercise in the upcoming pages, please do not close the document.