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Introduction

CATIA 3DEXPERIENCE 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

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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 3DEXPERIENCE. The exercises in this book will list steps for you to complete, along with explanations that try to inform you about 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 3DEXPERIENCE.

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 3DEXPERIENCE uses the Sketcher app as its principal method to create profiles. These profiles can be shaped and located via many different types of constraints. The first objective of the course is to learn how to use Sketcher and how to constrain profiles to the desired specifications. Sketcher is a very powerful method for creating profiles, and it is easy to use.

The second objective of the course is to use these sketches in Part Design. The sketches define two-dimensional cross-sections to be used for design three-dimensional shapes. There are several different shapes that can be made and 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 onto the sketch plane. It also includes 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 how to 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. This is a fairly simple process in CATIA 3DEXPERIENCE, and it is the real strength of Part Design.

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

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

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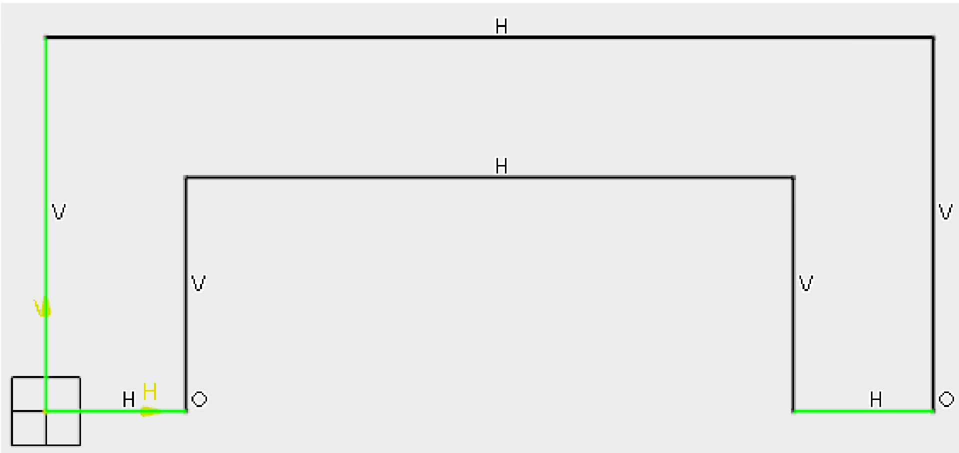
Profiles

This section will discuss the Profile icon. It is perhaps the most commonly used icon when defining sketches. The following exercises demonstrate the usefulness of the Profile icon, and how to utilize it effectively. It can create simple shapes and complex shapes within one operation.

The typical method of using the Profile tool is to specify the corner points of the desired shape. CATIA will generate lines between those points until you either select on the Profile icon again, double select a location in the workspace, or select the starting point to close the profile. You can also use sub-options that appear in the Sketch tools toolbar to generate curves as you are defining the profile. Alternatively, you can generate a tangent curve by using the first mouse button while sketching with the Profile icon. This will be done in a later exercise.

When defining a profile, the geometry may turn blue, or blue constraint symbols may appear. This indicates that a constraint will be created if a location is selected at that moment. It is useful for defining horizontal and vertical lines, or for defining tangencies with arcs and circles.

You will now build several profiles to practice using the various capabilities of the Profile icon. The first profile you are going to build looks like the one shown here.

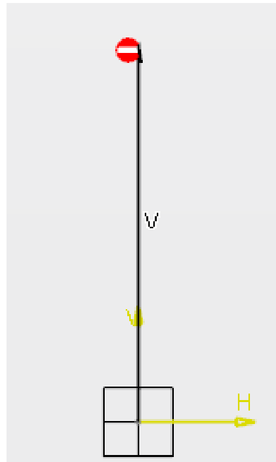


Select the Profile icon.  It should be highlighted.

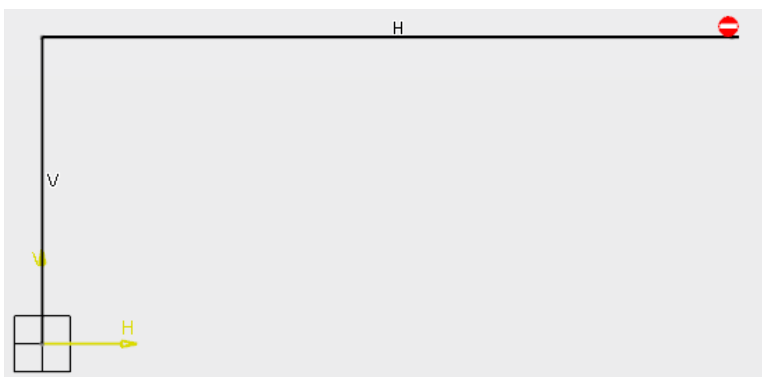
Select the origin point of the sketch plane. This specifies the starting point for your profile. When you specify the next location, make sure the line appears blue before selecting in the workspace. This will place a vertical constraint on your element automatically.

Note: If the automatic constraints become problematic while you are sketching, you can hold down the Shift key to temporarily ignore constraints. Once you release Shift, constraints will be recognized again.

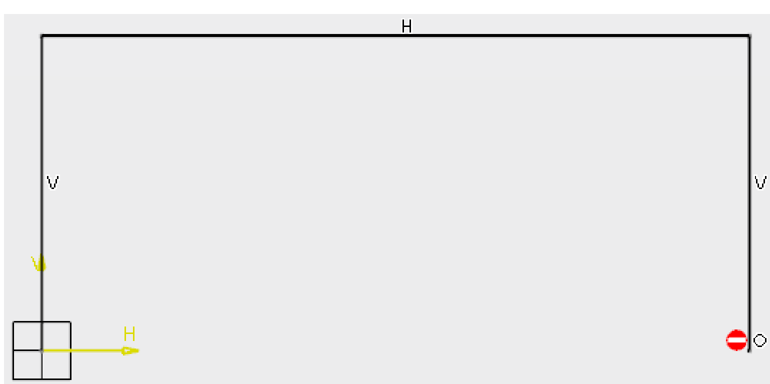
Select a location above the origin. If the line turned blue before you selected the second location, it should appear with the vertical constraint on it like the one shown below. Make sure when you specify the other locations that those elements also appear blue before selecting in the workspace.



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.

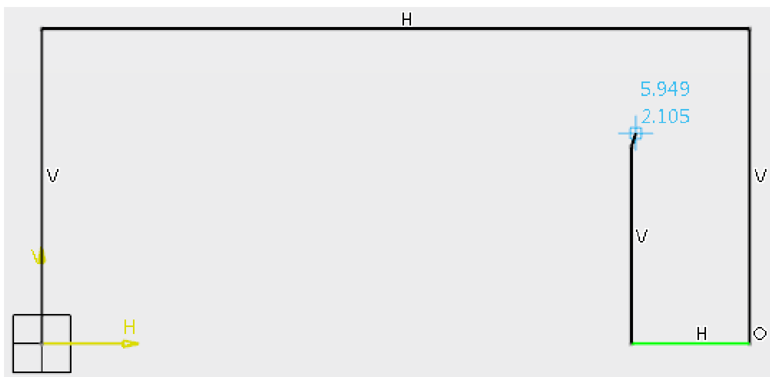


Select a location below the previous location and on the *H* axis. It should appear similar to the diagram shown below. You might notice a little, green circle appear. This is a coincidence constraint. This coincidence constraint forces the end point to be aligned with the *H* axis. This and other constraints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

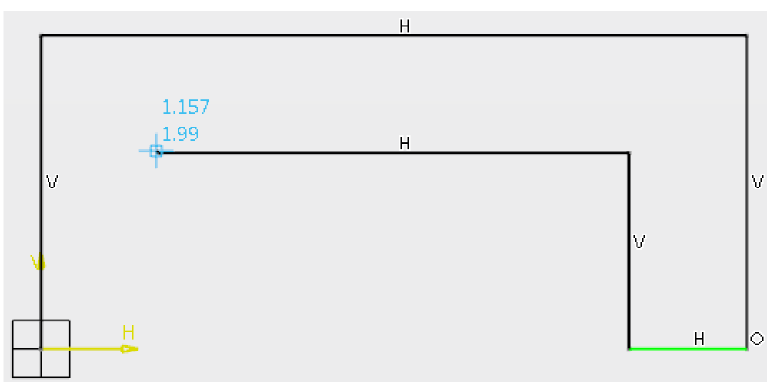


Select a location to the left of the previous location. You may have noticed that the first vertical line and the shorter horizontal line both turned green. This means they are iso-constrained. Constraints will be discussed with greater detail in later exercises.

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.



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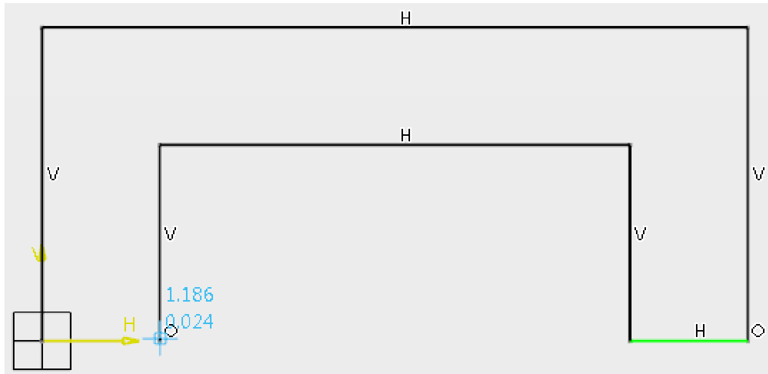
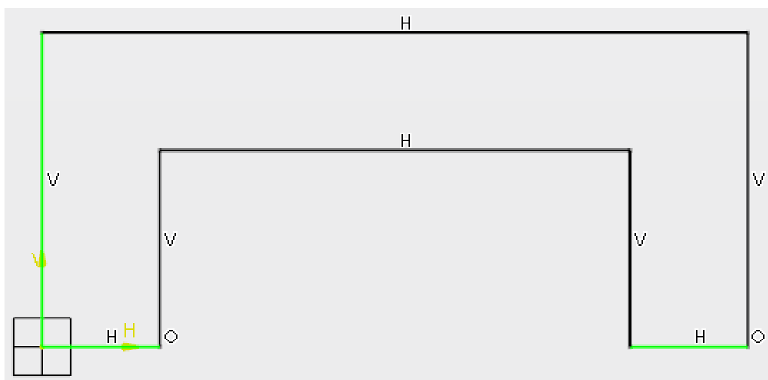
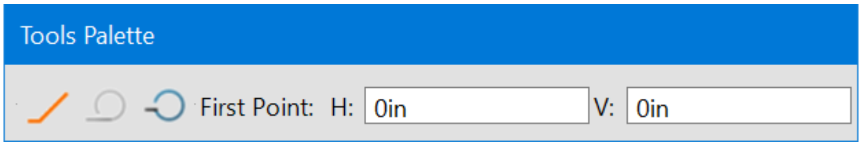
Select a location below the previous location. Your sketch should look similar to what is shown here.



Select the origin point of the sketch again. As long as you create the entire profile at one time, selecting the start point again will close the profile and end the command. You can undo selections in the middle of creating your profile by using the Undo icon, or by using the Ctrl Z keyboard shortcut.  The Undo icon is a standard icon so it is available on each of the toolbar sections.



The sub-options for the Profile icon are available within the *Tools Palette* toolbar and are shown below.



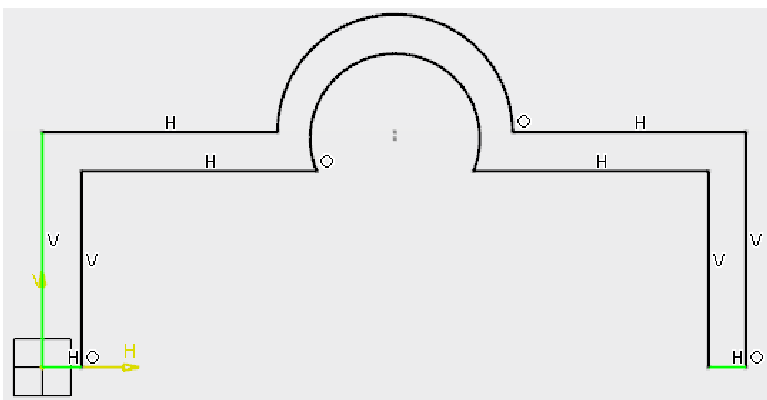
 Line

 Tangent Arc

 Three Point Arc

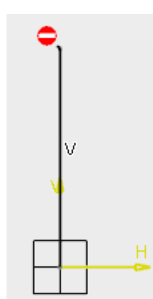
You will now perform a sketch with the Profile icon by using the Three Point Arc sub-option. It allows you to define an arc without having to exit the Profile command.

Select the Profile icon.  The next profile you are going to build looks like this.

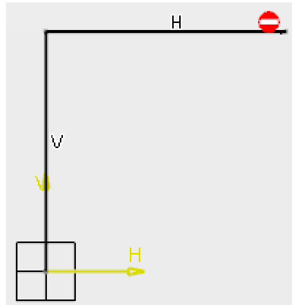


Select the origin point of the sketch plane. This specifies the starting point for your profile. When you specify the next location, make sure the line appears blue before selecting in the workspace. This will place a vertical constraint on your element automatically.

Select a location above the origin. If the line appeared blue before you selected the second location, 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 also appear blue before selecting in the workspace.



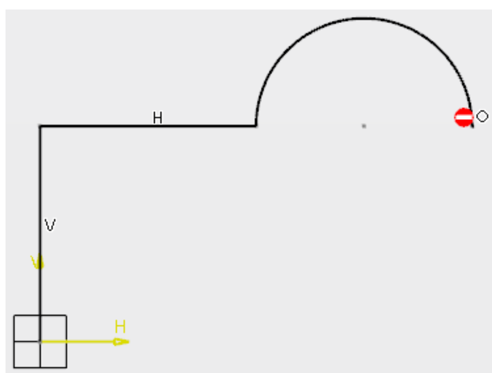
Select a location to the right of the previous location. It should appear similar to the diagram shown below.



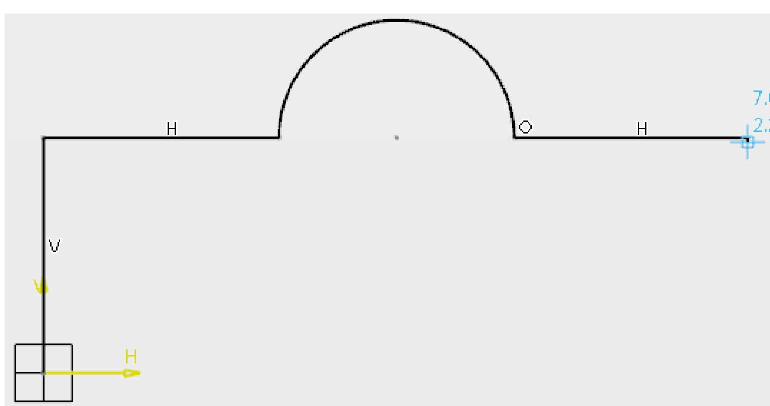
Select the Three Point Arc icon from the *Tools Palette* toolbar.  This icon will allow you to specify a location for the arc to pass through and a location for the arc to end at. The arc will begin at the last location specified, which, in this case, is the endpoint of the horizontal line.

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.

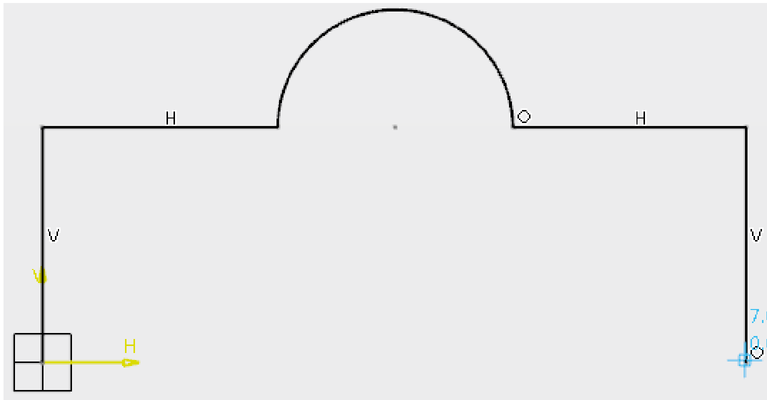
Select down and to the right of the previous location. This location should be straight across from the start of the arc. It should appear similar to the diagram shown below. Notice how the Three Point Arc icon in the Sketch tools toolbar automatically turned off and the Line icon turned back on.



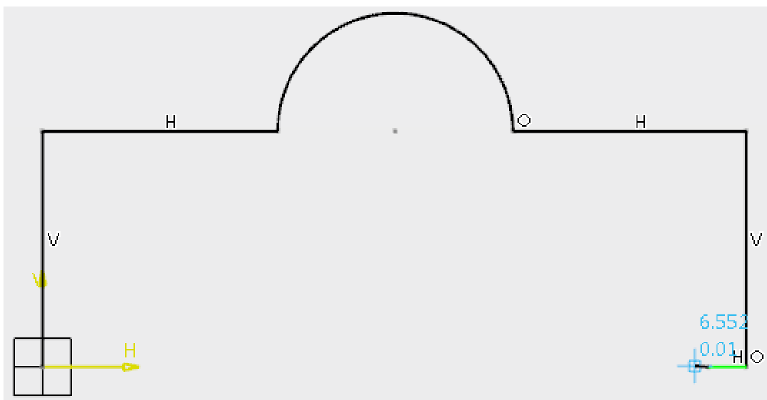
Select to the right of the previous location.



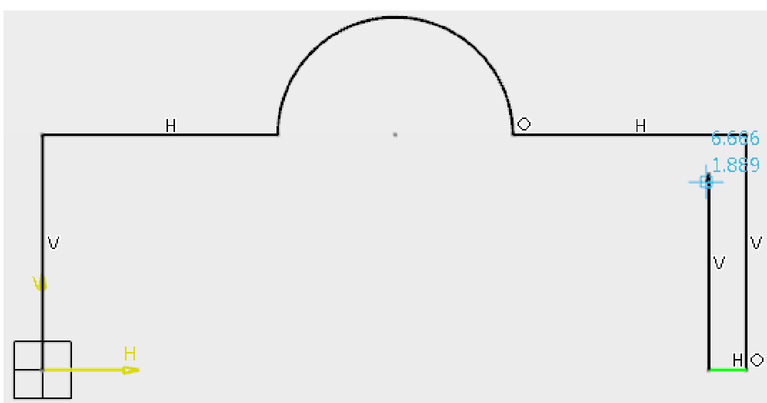
Select below the previous location, on the *H* axis. It should appear similar to the diagram shown below.



Select to the left of the previous location.



Select above the previous location, as shown below.



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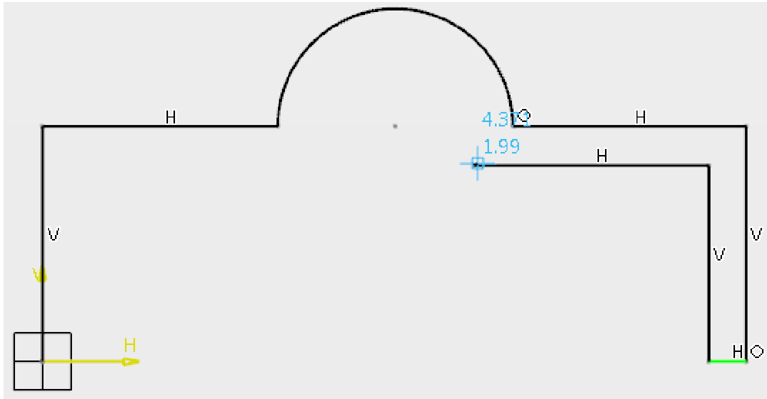
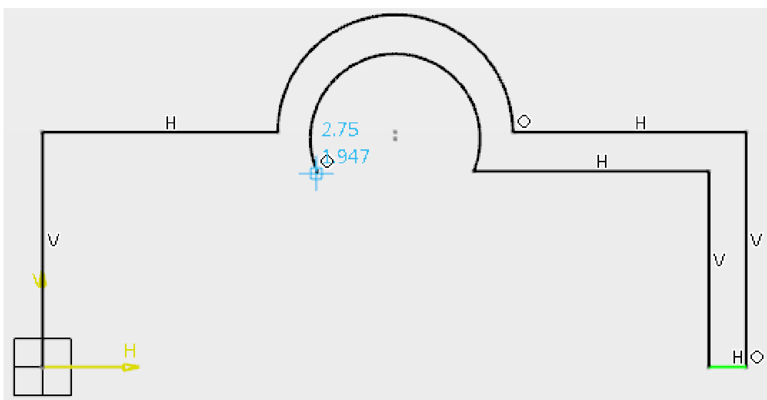
Select to the left of the previous location.

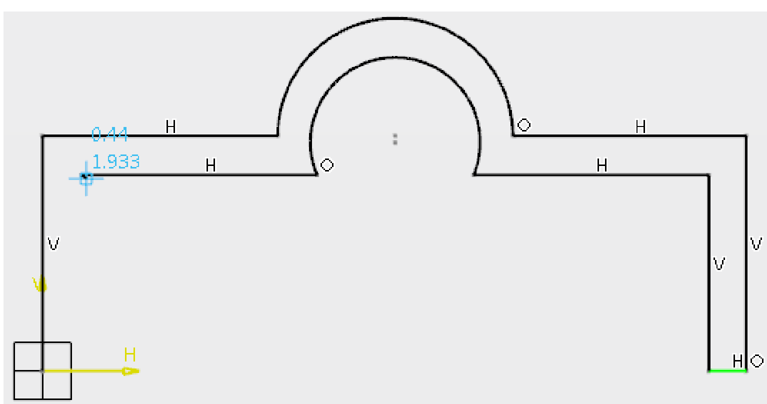


Select the Three Point Arc icon, then select up and to the left of the previous location.  This is the location that the arc will pass through. The next point specifies where the arc will end.

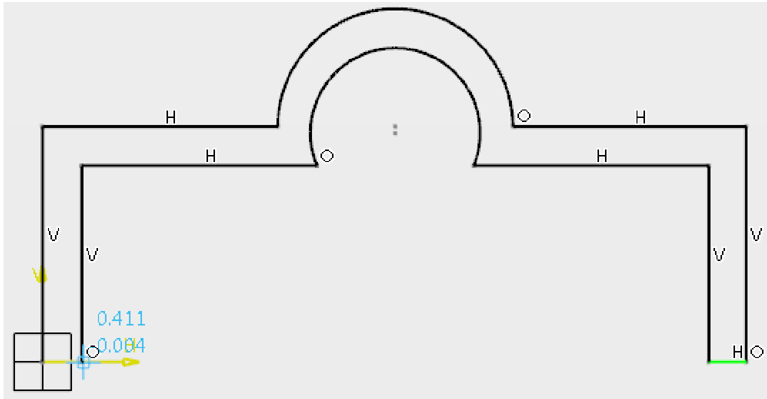
Select down and to the left of the previous location. This location should be straight across from the start of the arc. It should appear similar to the diagram shown below.



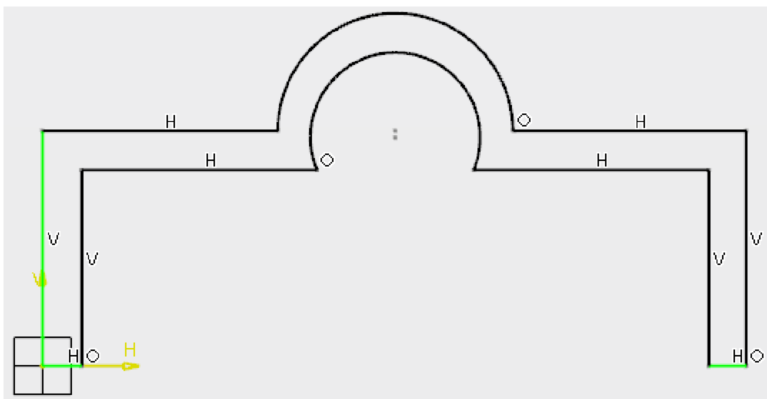
Select to the left of the previous location.



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



Select the origin point of the sketch again. CATIA closes the profile and exits the Profile command.



Basic Part Design

The following section will cover the basic use of the Part Design workbench to create parts. It will consist of three parts: basic shapes, operations on shapes, and interfacing between Part Design and Sketcher.

Basic Shapes

This part will discuss the various shapes that can be created by using the icons within the Part Design workbench. The purpose of the following exercises is to introduce how to use the icons and their options. Their usefulness depends on the part you are trying to create. It is important for you to understand how to use each of the tools in conjunction with your sketches in order to produce a final part.

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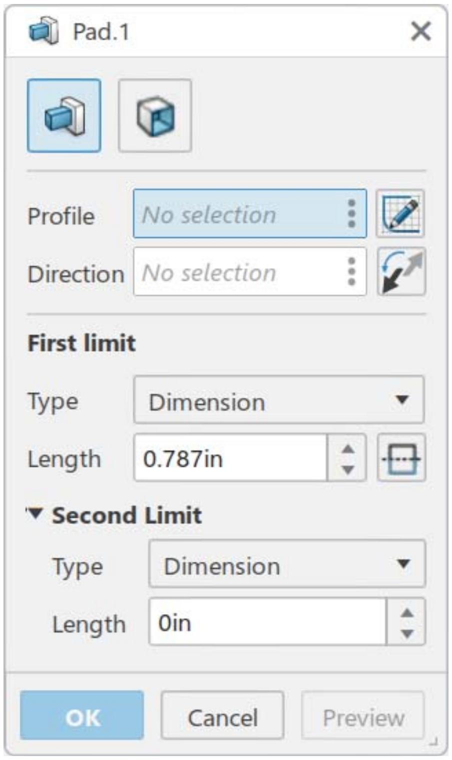
Pad

The **Pad** option allows you to use a sketch and extrude it in a linear direction to produce a solid pad. You can create a sketch or profile on-the-fly by pressing the third mouse button while in the *Profile* box. This allows you to use one of the available options to define a profile if you did not already have one created. When you create a pad, the *Pad* window appears as shown here.



Creates a standard linear extruded pad

Creates a thin feature pad where the selected profile can have thickness added

Profile Specifies which sketch will be used; you have the option to create or modify the sketch using the **Sketcher** icon next to the box. You can also select a surface to use as your profile.

Direction Specifies the direction for the pad to be extruded. By default if a planar profile is selected, it will extrude normal to the profile. However, a different direction can be selected. The direction can be inverted using the **Invert** icon next to the selection pane.

Initially, the window will appear with only the *First Limit* displayed. You can select the *Second Limit* to expand the options. Since the options are the same for both limits, they will only be discussed once.

Type

Dimension Allows you to enter a length value

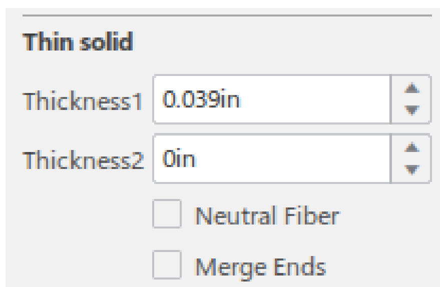
Up to next Extends to the next feature of an existing part

Up to last Extends to the last feature of an existing part

Up to plane Extends to a specified plane, which is its *Limit*

Up to surface Extends to a specified surface, which is its *Limit*

Length Specifies the length of the extrusion. You can specify the extrusion will extrude the same distance in both directions by selecting the Mirror icon next to the input pane.



Thin solid If the Thin Feature icon is selected, the *Thin solid* options will appear.

Thickness1/2 Specifies the wall thickness that will be applied to each sketch element

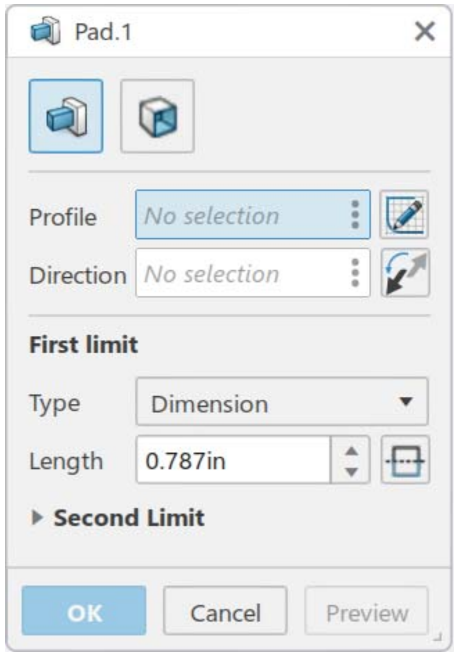
Neutral Fiber Forces the sketch element to be in the center; the wall thickness is added to both sides equally

Merge Ends Extends or trims the elements to existing material

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

Open the PDAS - Pad1 document. There are two sketches already created for you.

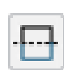
Select the Pad icon.  You will be able to create a pad using one of the sketches. This exercise is going to cover the various methods that you can use to create pads. A *Pad* window should appear similar to the one shown below.

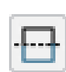


Select *Sketch.1* from either the graphical area or the tree. This will be the profile for the pad feature. You are going to use the basic option of keying in a length for the *First Limit*. You will also preview what the Mirror and Invert icons do.

Change the *Length* to 4.0. Do not press Enter yet. Doing so will automatically create the pad with the specified value. For now, you are going to want to *Preview* the pad in order to see what CATIA is doing until you understand the different options.

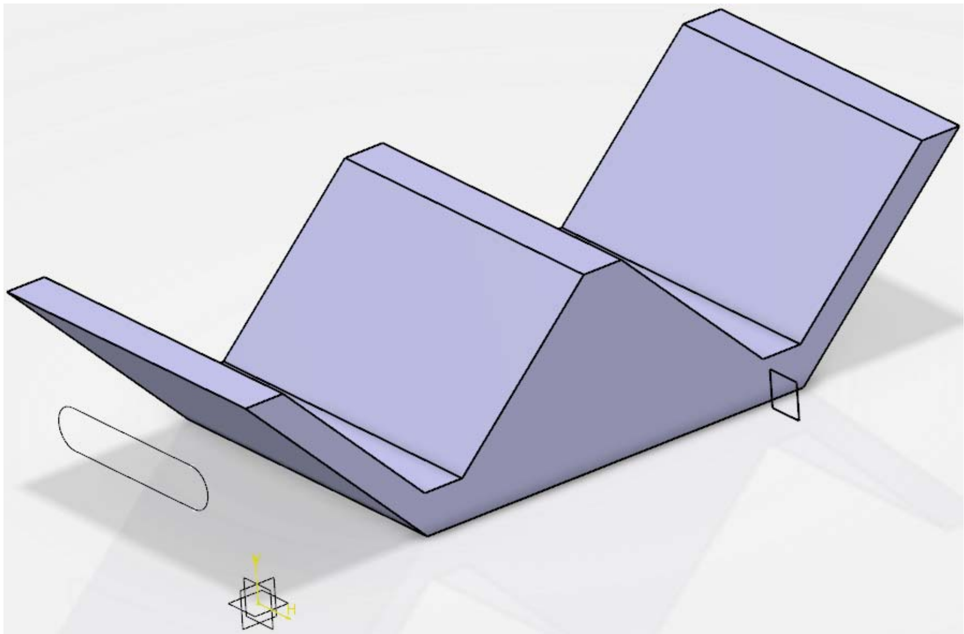
Select *Preview*. A preview of the pad appears. You will now change some of the other options to see the differences between them.

Select the Mirror icon, then select *Preview*.  As you can see, instead of the pad extending in only one direction, it now extends four inches on both sides. Your sketch is being used as the mirror plane.

Select the Mirror icon again to turn it off, then select *Preview*.  Now you are going to reverse the direction of the pad so that it will extend in the opposite direction only.

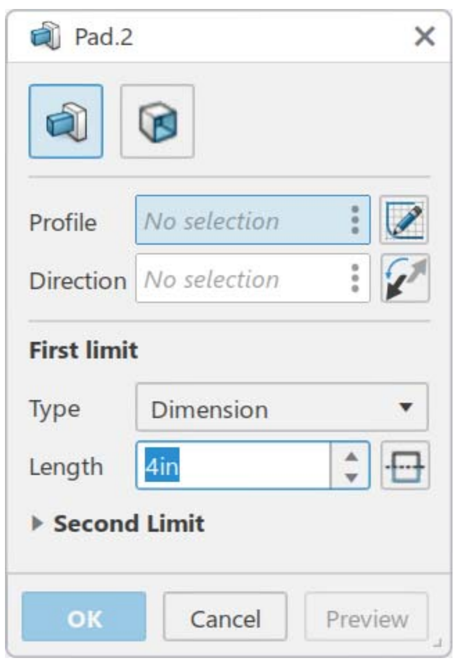
Select the Invert icon, then select *Preview*.  The pad is still going to be four inches wide. This is the side you want to create the pad on.

Select *OK*. The pad should appear similar to the diagram shown below. The sketch was automatically hidden after being used by the pad. This is due to a setting under the pull down menu *Preferences*.



You are now going to explore the other *Types* you can use to define limits for your pads.

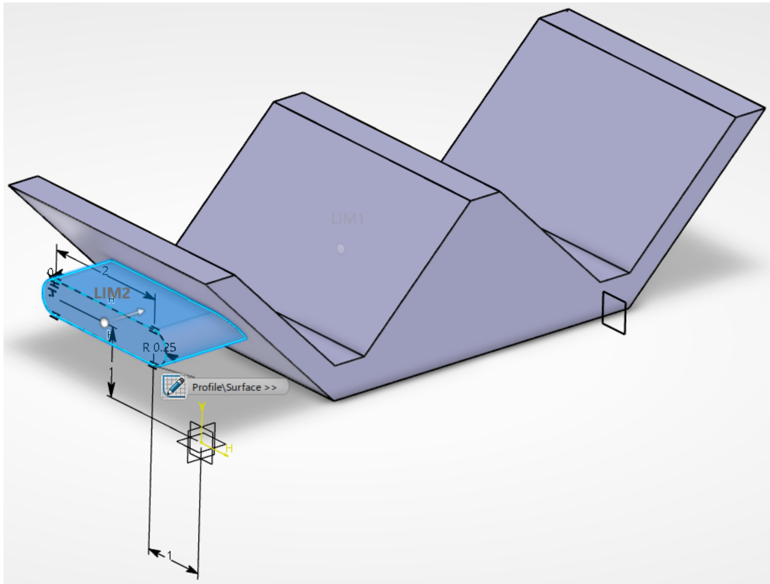
Select the Pad icon.  The *Pad* window appears.



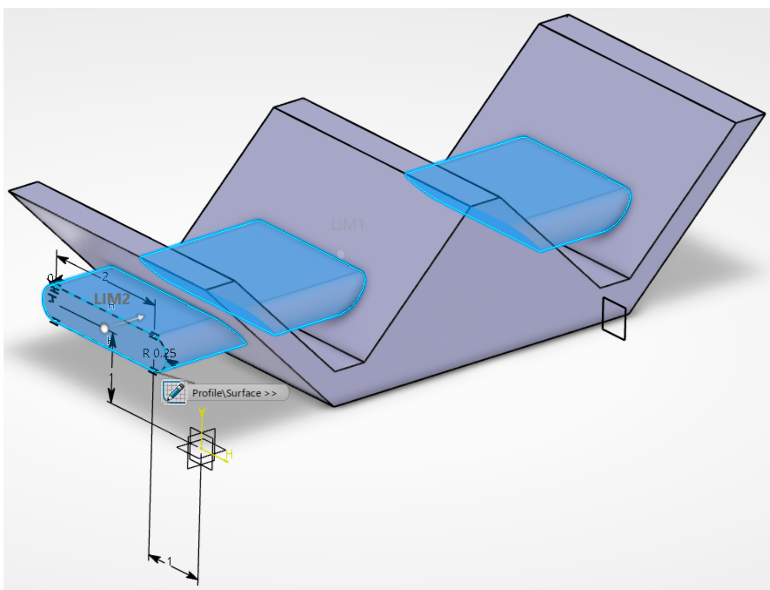
Select *Sketch.2*. This specifies the sketch you want to use for the next pad.

Select the Invert icon, so that it extends 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.

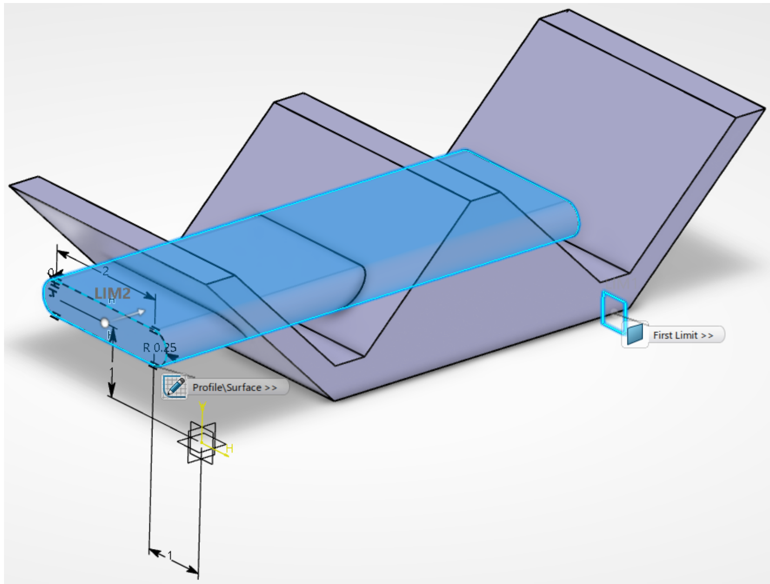


Change the *Type* to *Up to last* and select *Preview*. Notice that the pad goes all the way to the last side of the previous part. It should appear similar to the diagram shown below.



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.

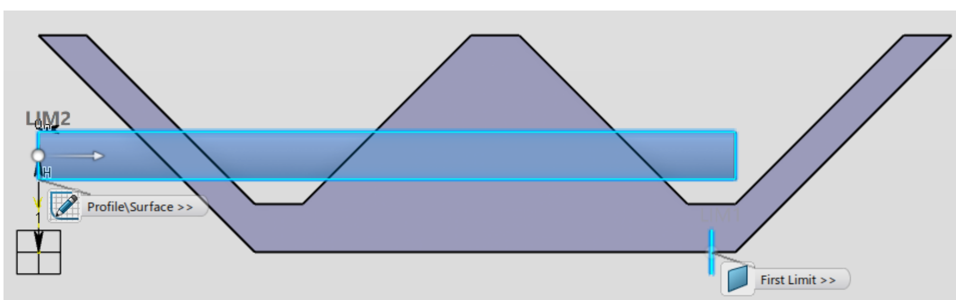
Select the plane indicated below, then select *Preview*. Notice that the pad goes up to the plane and then stops. It should appear similar to the diagram shown here.



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.

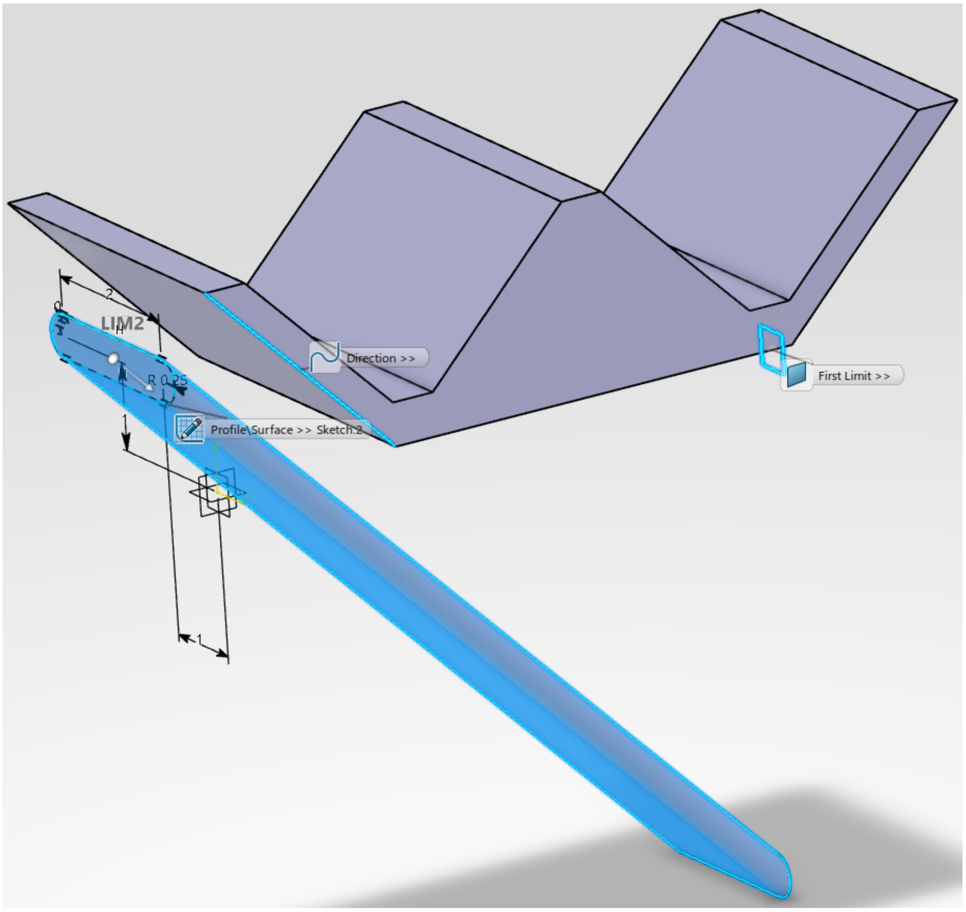
The *Offset* field is now available, and you are able to enter both positive and negative values. A positive value will extend the pad beyond the limit by the specified amount, whereas a negative value will stop the pad short of the limit by the specified amount.

Enter 0.25 for the *Offset*, then select *Preview*. From the side, you can see that the pad extends past the selected limit plane.



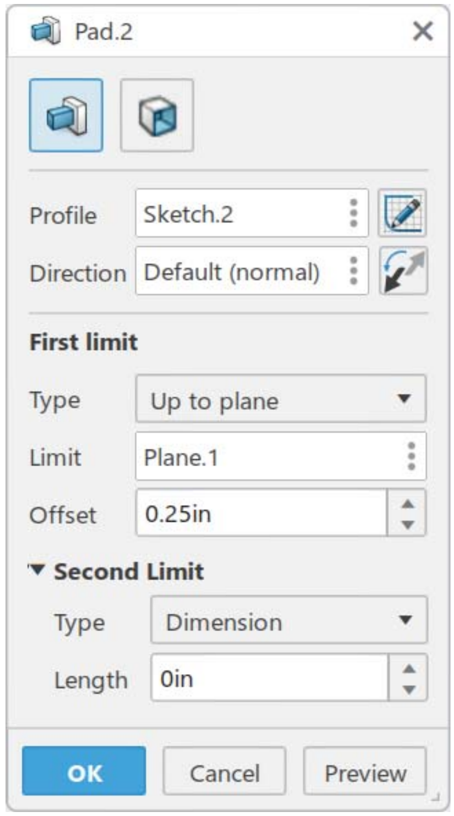
Currently, the *Direction* defaults to be normal to the profile. You will now specify an element to be used for the direction instead.

Select in the *Direction* field, then 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 specified earlier. It should appear similar to the diagram below.

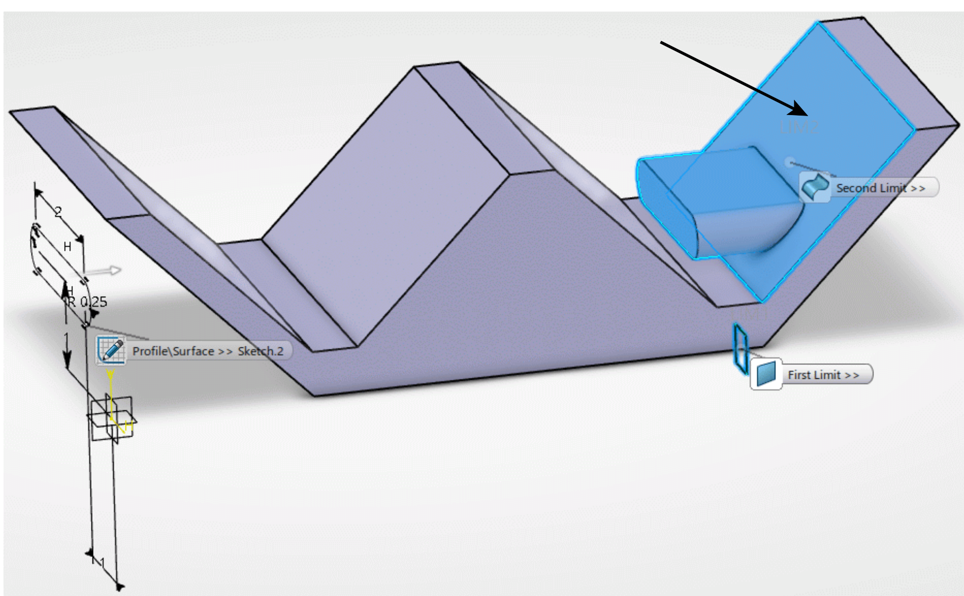


Right select in the *Direction* pane and select *Default Selection* from the contextual menu, then select *Preview*. The direction is once again normal to the sketch plane. You will now use a *First Limit* and a *Second Limit* together to create the pad.

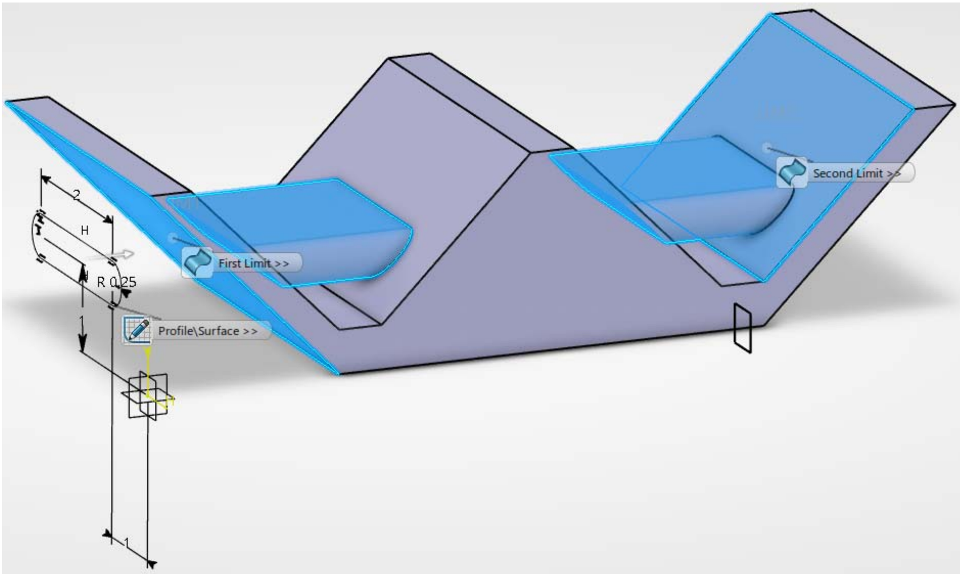
Select the *Second Limit* to expand the options. This expands the window to reveal more options. Your window should appear similar to this.



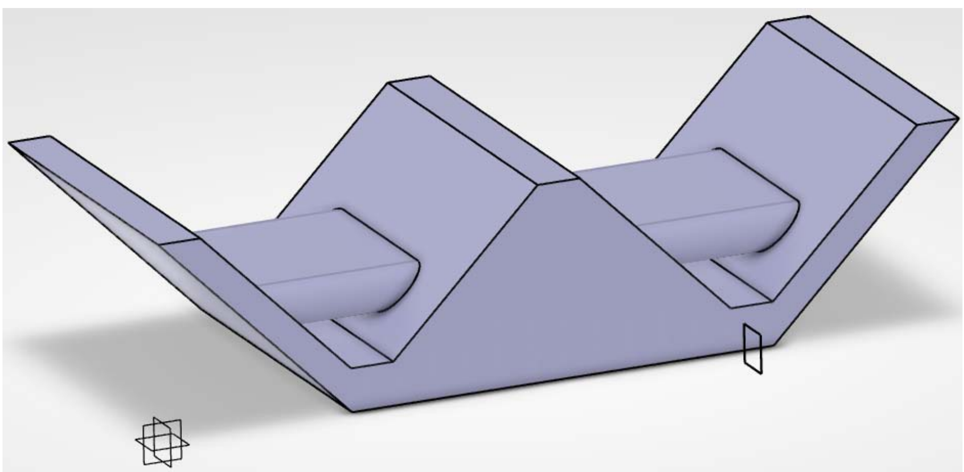
Under the *Second Limit*, change the *Type* to *Up to plane*, then select the face indicated below and click *Preview*. Your pad will now exist between the plane selected for the *First Limit* and the selected face for the *Second Limit*, plus the offset value. You can use faces for the *Up to Plane* option as long as they are planar. You can also use planes for the *Up to Surface* option.



Change the *Type* for the *First Limit* to *Up to Surface* and select the face closest to *Sketch.2*, then enter 0.0 for the *Offset* and select *Preview.* Your pad should appear as depicted below.



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



This exercise showed most of the options available when creating a pad. There are other shapes you will see that have some of the same options. Hopefully, you have a good understanding of what each option allows you to do.

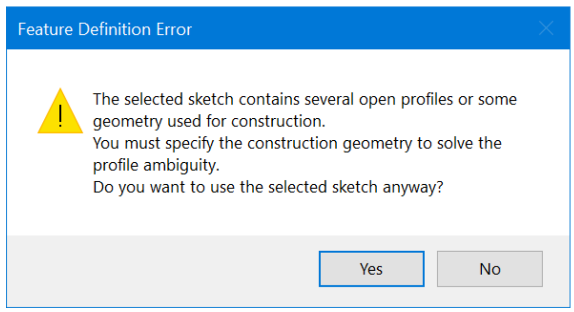
Note: Open profiles (sketches) can be used to create pads or pockets, as long as they will be closed by the other faces of your existing part. You will see this demonstrated in the next exercise.

Save and close the document.

Open the PDAS - Pad2 document. A sketch has already been created. 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*. A *Feature Definition Error* window appears. It is because your sketch contains open profiles. However, this is okay since you will be using the *Thick solid* option.



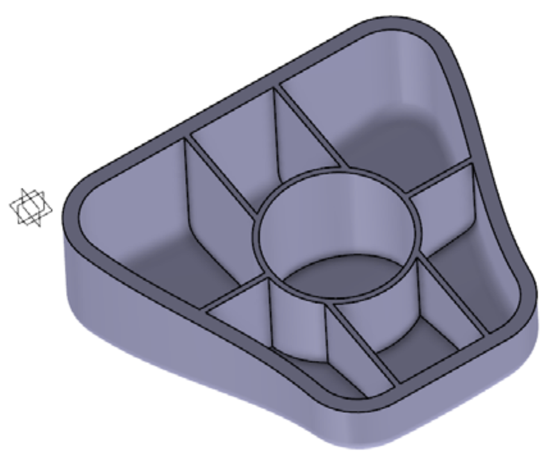
Select *Yes*.

Select the Thin Feature icon.  The *Thin solid* options appear. Whenever you are using an open profile, you will want to select the *Thick solid* option before doing anything else. Otherwise, you will continue to receive error messages until it is turned on.

Turn on *Neutral Fiber* and enter 0.1 for *Thickness1*. This option splits the specified thickness in half, distributing the solid evenly in both directions. Make sure the directional arrow in the graphical area is pointing toward the existing solid. If it is pointing away, select the Invert icon.

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

Select *OK*. Your part should appear similar to the diagram shown below.

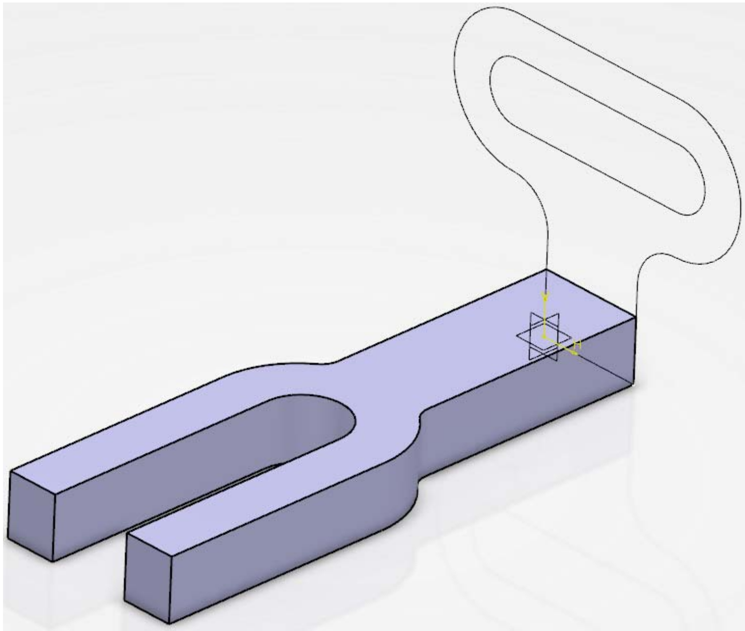


Save and close the document.

Open the PDAS - Pad3 document. There are three sketches already created for you.

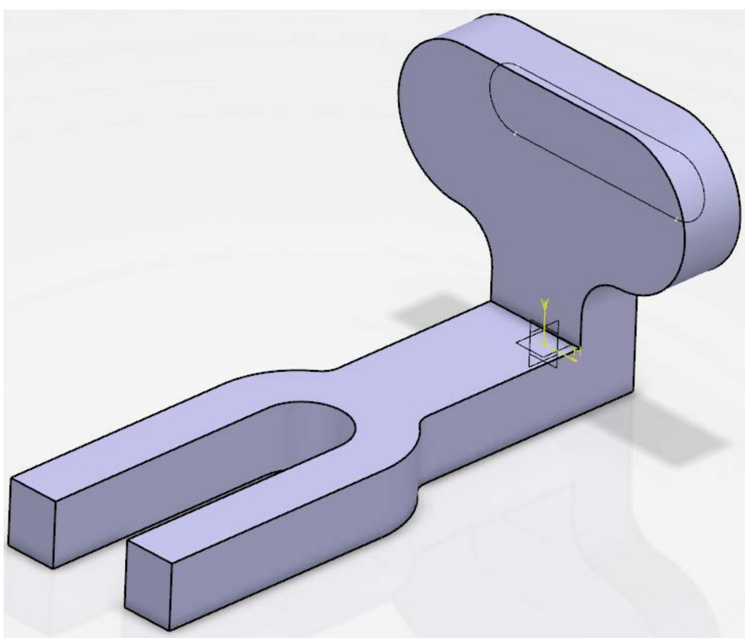
Select the Pad icon.  The *Pad* window appears.

Set the *Type* to *Dimension* with a *Length* of 0.75, then select *Sketch.1* and click *OK.* The pad should appear similar to the diagram shown below.



Select the Pad icon again, then select *Sketch.2*. 

Set the *Type* to *Dimension* with a *Length* of 0.75, then select *OK.* Your part should look like this.



Do not close the document; you will continue to use it for the next exercise.

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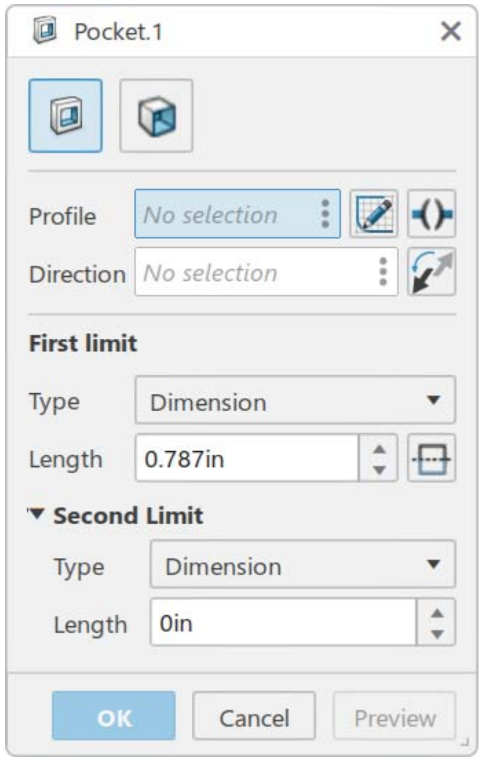
Pocket

The Pocket tool allows you to use a sketch to remove material in a linear direction, thereby producing a pocket. You can create a sketch or profile on-the-fly by pressing the third mouse button while in the *Profile* box. When you select the icon, a *Pocket* window appears like the one shown below.



 Creates a standard linear extruded pocket

 Creates a thin feature pocket where the selected profile can have thickness added

Profile Specifies which sketch will be used; you have the option to create or modify the sketch using the Sketcher icon next to the box. You can also select a surface to use as your profile.

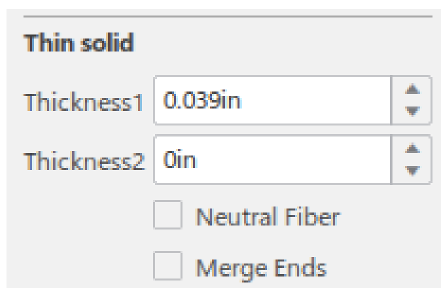
Direction Specifies the direction for the pocket to be extruded. By default if a planar profile is selected, it will extrude normal to the profile. However, a different direction can be selected. The direction can be inverted using the Invert icon next to the selection pane.

Initially, the window will appear with only the *First Limit* shown. You can select the *Second Limit* to expand the options. Notice that these options are exactly the same as the Pad icon's options. The major difference between a pad and a pocket is that a pad adds material to your part, while a pocket removes material from your part.

Type

Dimension	Allows you to enter a *Length*
Up to next	Extends to the next feature of an existing part
Up to last	Extends to the last feature of an existing part
Up to plane	Extends to a specified plane, which is its *Limit*
Up to surface	Extends to a specified surface, which is its *Limit*

Length Specifies the length of the extrusion. You can specify the extrusion will extrude the same distance in both directions by selecting the Mirror icon next to the input pane.



Thin solid If the Thin Feature icon is selected, the *Thin solid* options will appear.

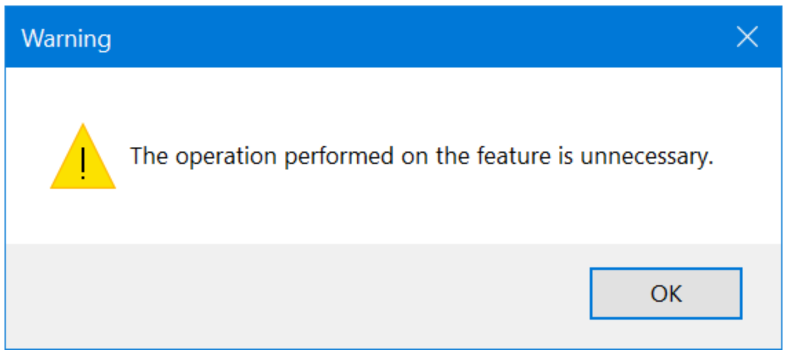
Thickness1/2	Specifies the wall thickness that will be applied to each sketch element
Neutral Fiber	Forces the sketch element to be in the center; the wall thickness is added to both sides equally
Merge Ends	Extends or trims the elements to existing material

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

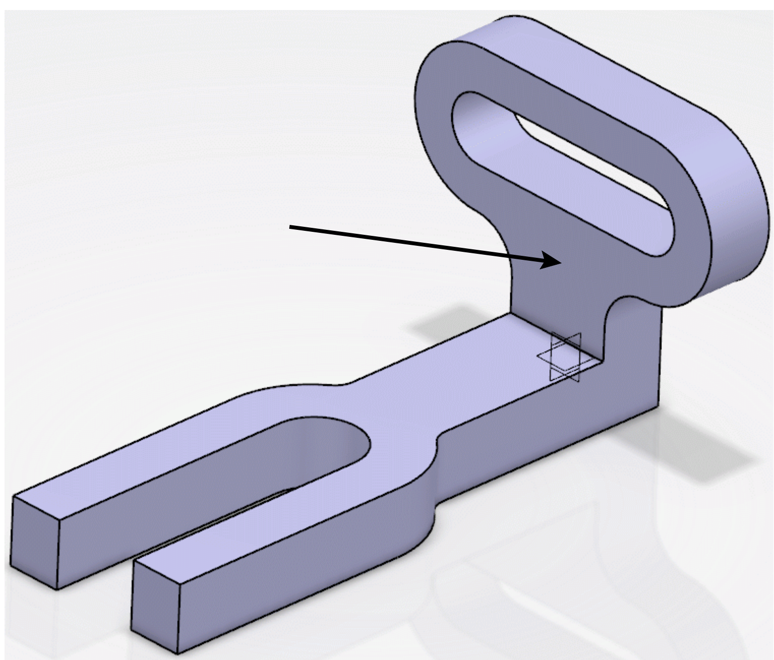
You will now create a pocket in the existing part using *Sketch.3*.

Select the Pocket icon.  It is located in the sub-toolbar of the Pad icon. This will allow you to create a pocket using one of the sketches.

Select *Sketch.3*, then select the Invert icon.  You need to pocket toward the existing solid. If you attempt to pocket into space without removing any material, the following *Warning* message will appear.



Set the *Type* for *First Limit* to *Up to Next* and select *OK*. The pocket should appear similar to the diagram shown below.



By using *Up to Next*, you are ensuring that the pocket will always update with the face indicated above. Should *Pad.2* be modified to have a greater *Length* value, the pocket will still extend all the way through the pad because of the link to that face.

Save and close the document.