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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 also need to do a lot of reading 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 of 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 learn what you need 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 how to use the Sketcher workbench and constrain 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 it is also easy to use.

The second objective of the course is to use these sketches in Part Design. The sketches are two-dimensional cross-sections to be used for designing three-dimensional shapes. There are several 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 onto sketch planes. You will use basic formulas to set up typical values at multiple locations, as well as 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.

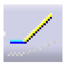
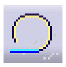
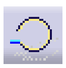
The fourth objective is to become efficient at modifying your designs. You can modify either by changing the parameters of a part operation, or by editing the sketch that was used. This is a fairly simple process in CATIA Version 5, and it is the real strength of Part Design.

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

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

Profile

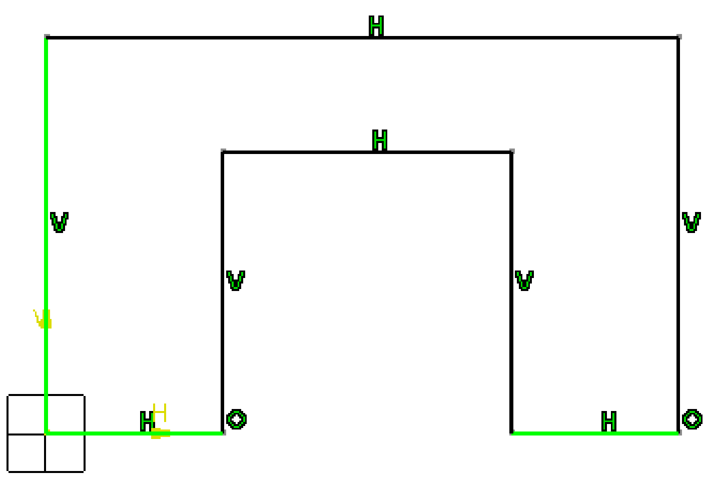
The Profile tool is one of the more commonly utilized icons when defining sketches. It creates shapes by connecting arcs and lines through a series of user-selected points. Both simple and complex shapes can be created in the same operation. Upon selecting the Profile tool, additional icons appear in the Sketch tools toolbar.

	Line	Creates a straight line segment
	Tangent Arc	Creates an arc that is tangent to the previous element within the profile
	Three Point Arc	Creates an arc starting at the current point, followed by a passing point and an end point

The Profile command will remain active until it is terminated in one of four ways: 1) double-selecting in space, 2) re-selecting the Profile icon, 3) pressing the Esc key, or 4) closing the shape by re-selecting its start point; this will only end the command if the entire shape is created in one operation.

While sketching with the Profile tool, thin, blue lines and blue, geometric constraints will occasionally appear. If a location is selected when blue constraints are visible, those same constraints will be created after the profile operation is complete. The thin, blue, solid lines indicate that coincidence constraints will be created. The thin, blue, perforated lines indicate that elements are aligned, but no constraints will be created.

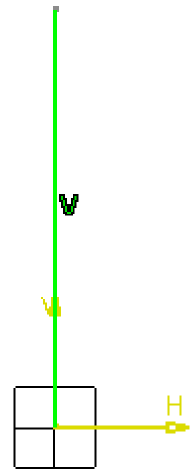
The following exercises will step through creating various shapes using the Profile tool. The first profile to be demonstrated is shown below.



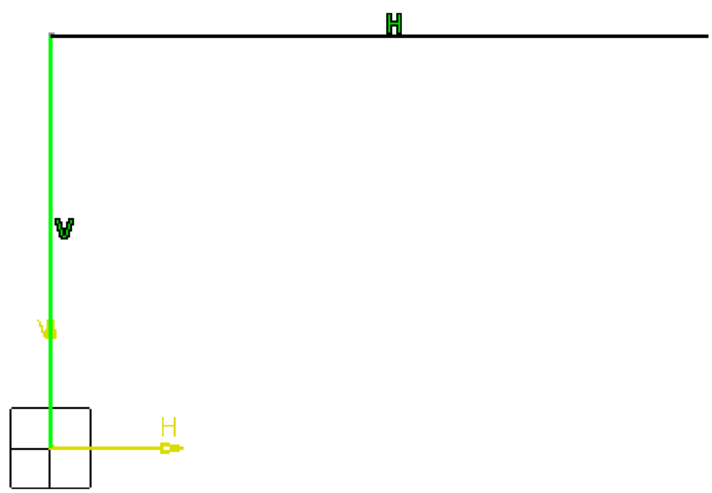
Select the Profile icon. 

Select the origin point of the sketch plane. This defines the start point of the profile.

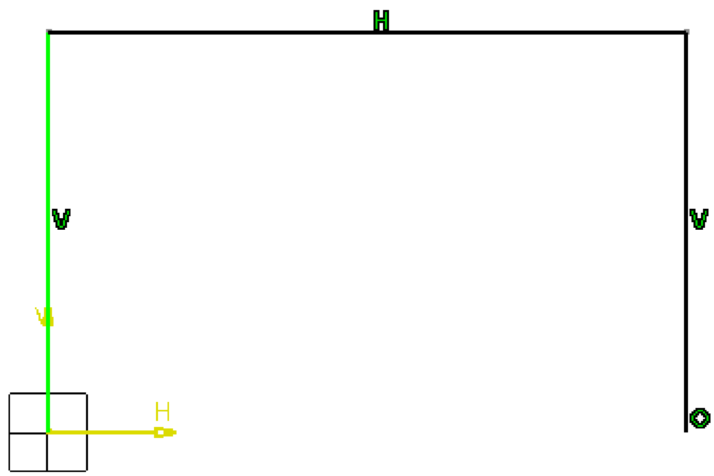
Select a location straight above the origin. Ensure the line is blue before selecting the next point. Doing so will create a vertical constraint.



Select straight to the right of the previous location. Ensure the line is blue before making the next selection. It should appear similar to below.

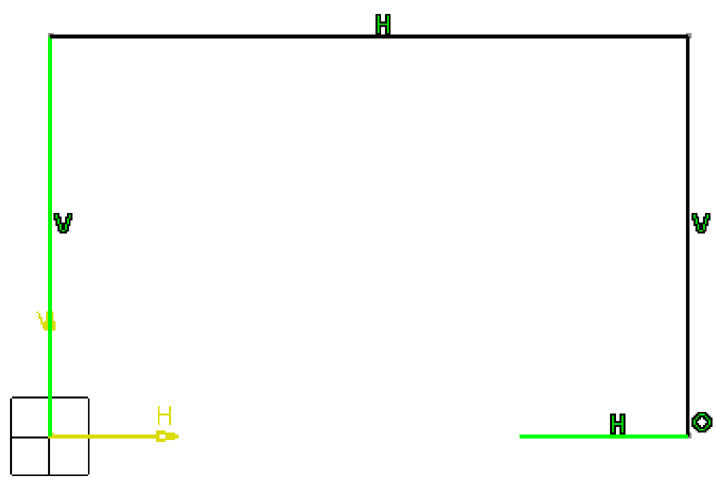


Select straight below the previous location and on the *H* axis. A vertical constraint and a coincidence constraint are created. The coincidence constraint is between the end point of the new line and the sketch's *H* axis.

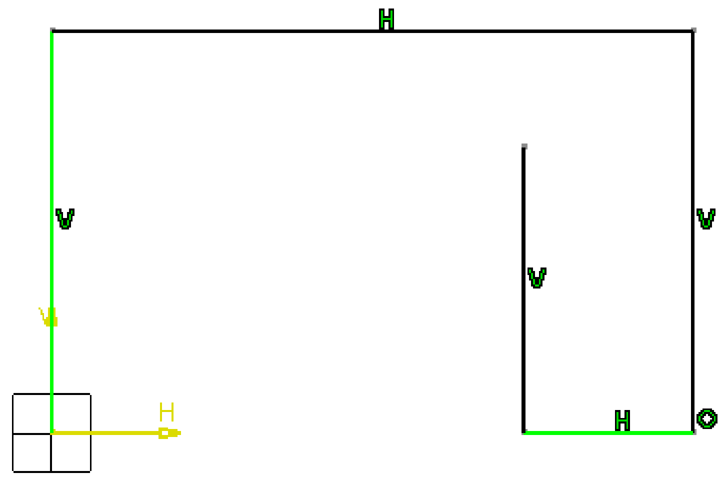


Note: At any point during the profile's creation, selecting the Undo *icon or* Ctrl+Z *will undo the previous selection. You can undo as many selections in a row as necessary.*

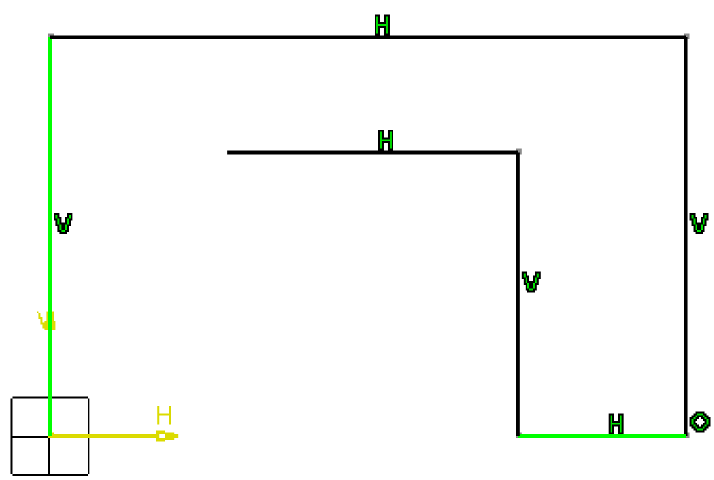
Select a location straight to the left of the previous end point. A horizontal constraint is created, and the line turns green. When an element is fully green, it is iso-constrained and cannot be moved. This will be discussed more later.



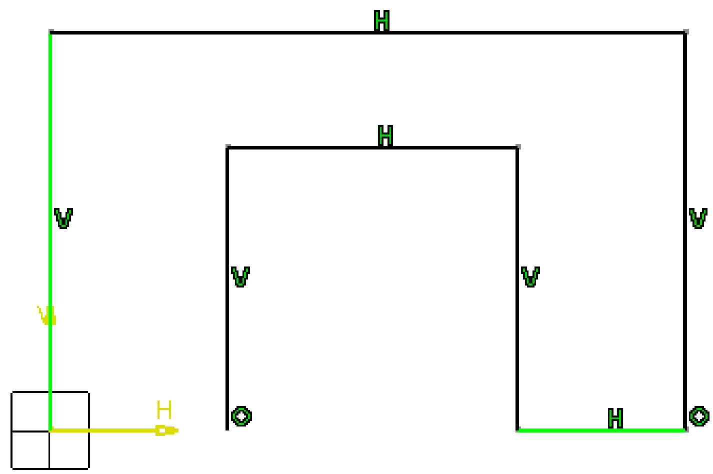
Select a location straight above the previous point. It should appear similar to below.



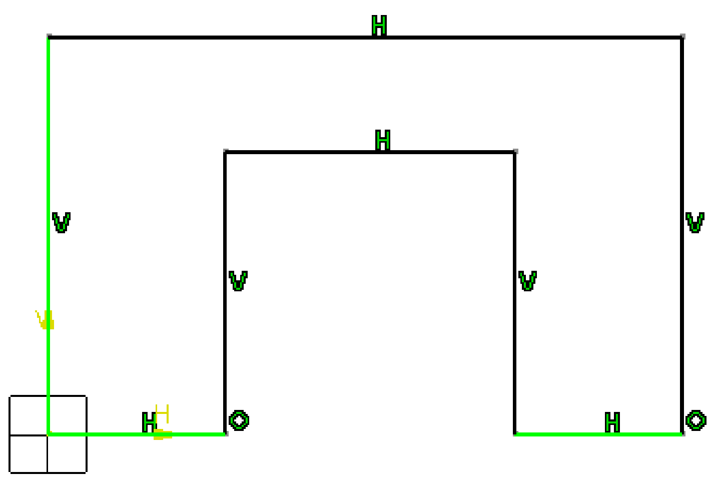
Select a location straight to the left of the previous location.



Select a location straight below the previous point and on the *V* axis.

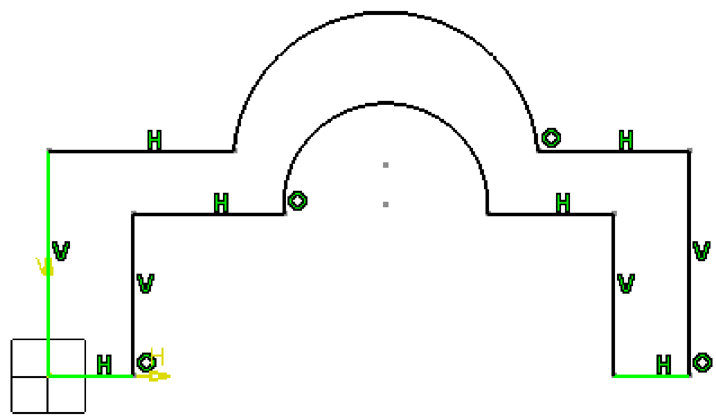


Select the sketch origin again for the last point. By selecting the same location for the start and end points in a single operation, the Profile command is automatically exited. The sketch should appear similar to below.



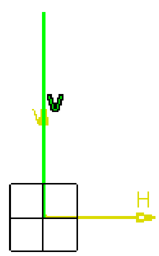
Save the document as <u>Profile1</u> and close it.

The second profile to be demonstrated is shown below. It will utilize the Three Point Arc tool in the Sketch tools toolbar.

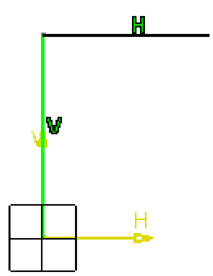


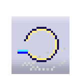
Select the Profile icon, then select the origin point of the sketch plane.  This defines the start point of the profile.

Select a location straight above the origin. For all line segments in this profile, ensure the line is blue before selecting the next point.



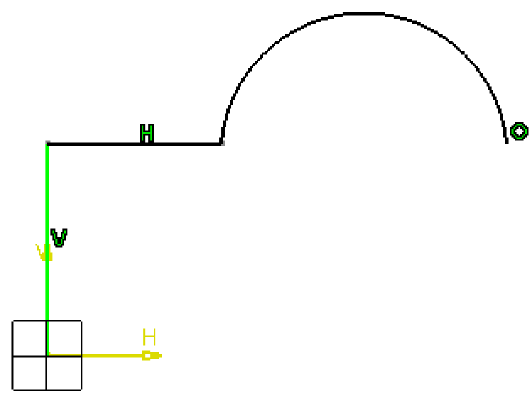
Select a location straight to the right of the previous point. It should appear similar to below.



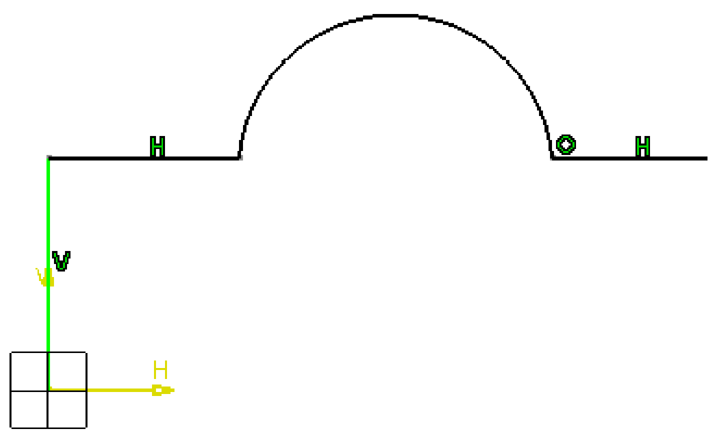
Select the Three Point Arc icon in the Sketch tools toolbar.  The arc will begin at the last profile location specified.

Select up and to the right of the previous location. This defines a point that the arc will pass through.

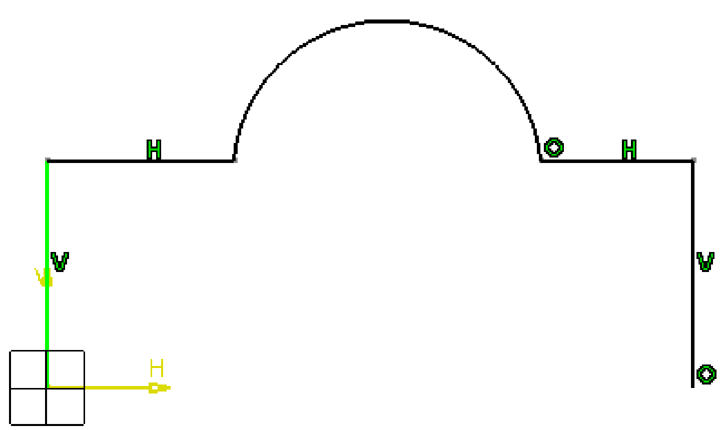
Move the mouse cursor down and to the right of the previous location, then click in space when you see a thin, blue, solid line appear. This location should be straight across from the start of the arc. Once the three point arc is complete, the Line icon in the Sketch tools toolbar is automatically selected. A coincidence constraint is created between the last point of the arc and the second profile line. It should appear similar to below.



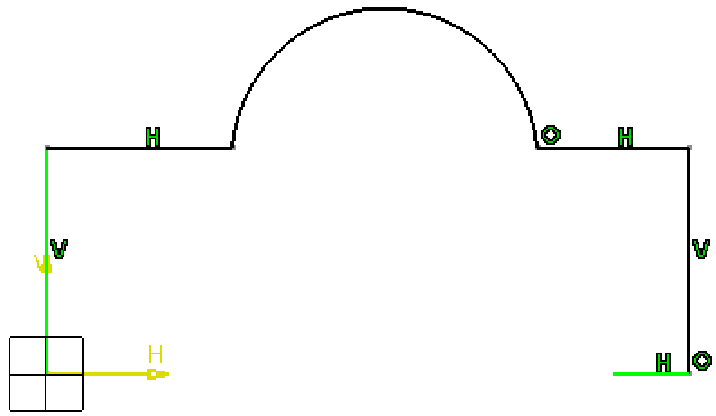
Select straight to the right of the previous location.



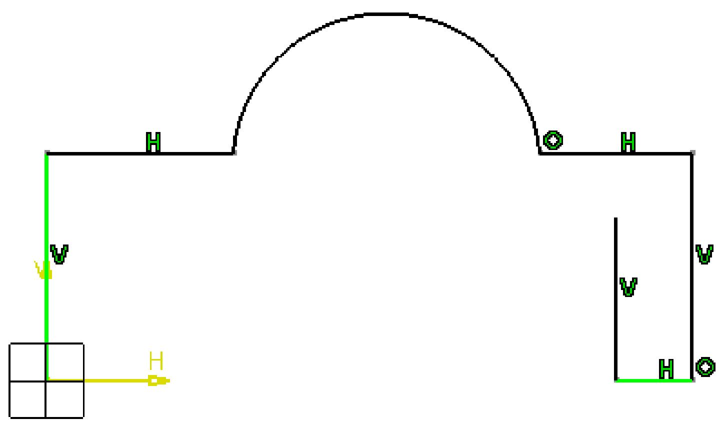
Select straight below the previous point and on the *H* axis.



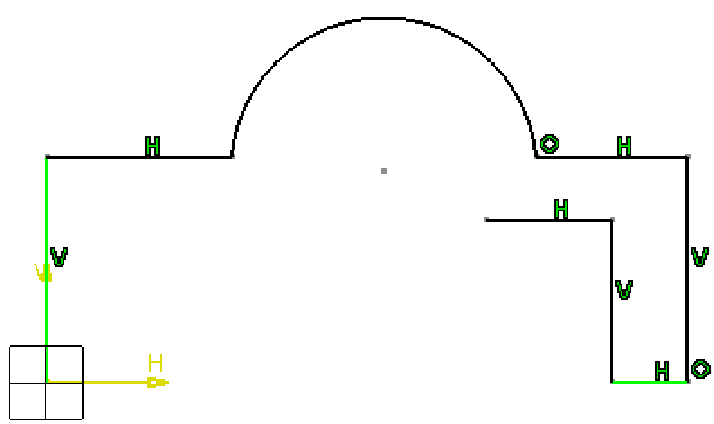
Select straight to the left of the previous location.

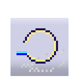


Select straight above the previous location.



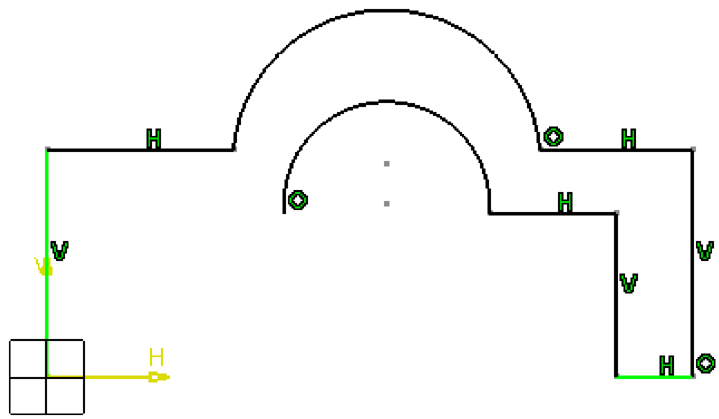
Select straight to the left of the previous location. It should appear similar to below.



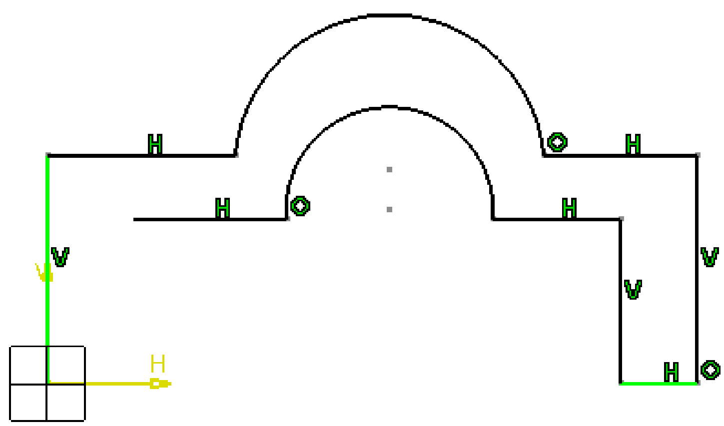
Select the Three Point Arc icon in the Sketch tools toolbar.  The arc will begin at the last profile location specified.

Select up and to the left of the previous location. This defines a point that the arc will pass through.

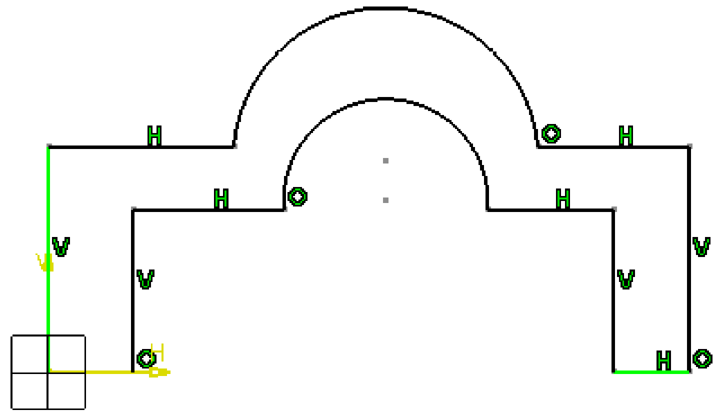
Move the mouse cursor down and to the left of the previous location, then click in space when you see a thin, blue, solid line appear. This location should be straight across from the start of the arc. Once the three point arc is complete, the Line icon in the Sketch tools toolbar is automatically selected. A coincidence constraint is created between the last point of the arc and the last line segment. It should appear similar to below.



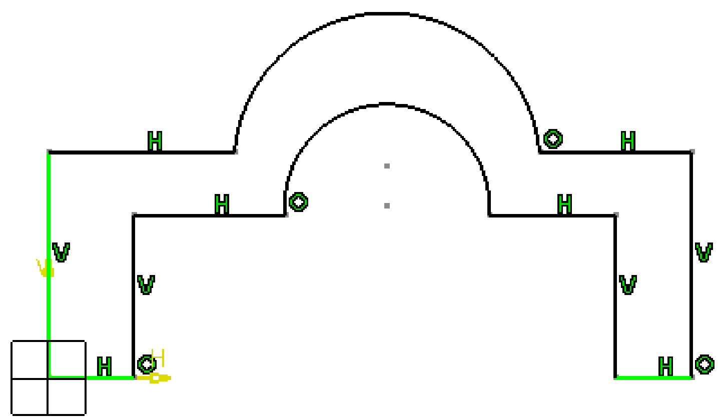
Select straight to the left of the previous location.



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



Select the sketch origin again for the last point. By selecting the same location for the start and end points in a single operation, the Profile command is automatically exited. The sketch should appear similar to below.



Save the document as <u>Profile2</u> and close it.

Basic Part Design

The next section covers the basic use of the Part Design workbench. It consists of three parts: basic shapes, operations on shapes, and interfacing between Part Design and Sketcher.

Basic Shapes

Many shapes can be created using the Part Design icons. The following exercises will demonstrate the use of these icons and their various options. The usefulness of each icon depends upon the part being created. It is important to understand how to use each tool in conjunction with sketches to produce a final part. The first tools to be discussed are found in the Sketch-Based Features toolbar.

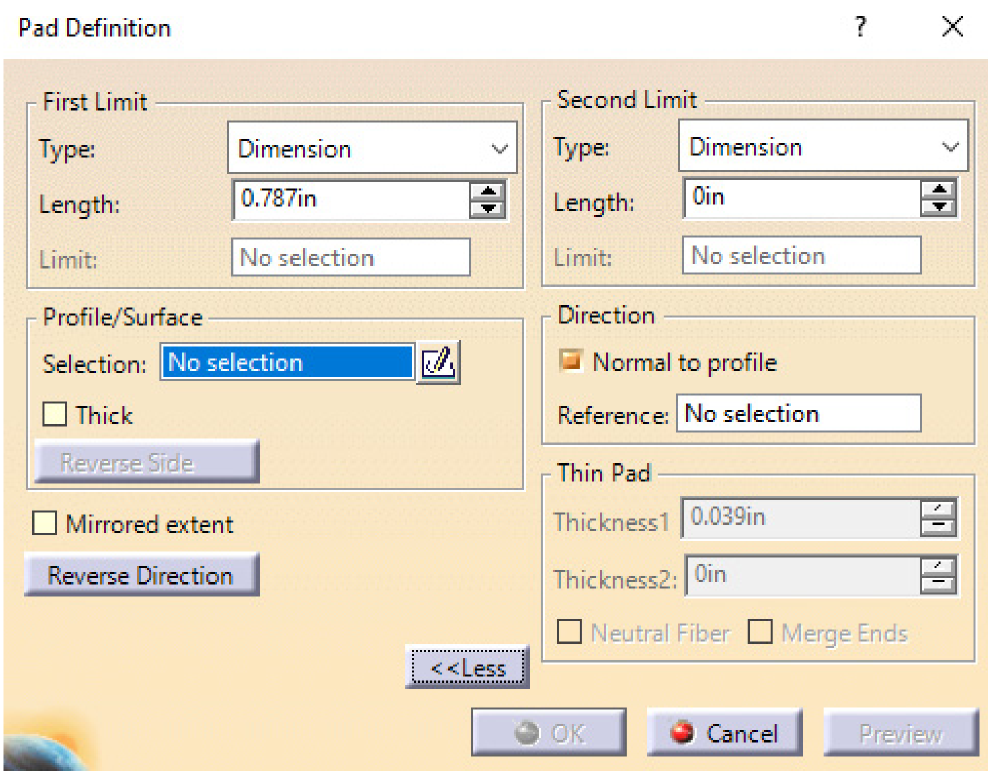
Pad

The Pad tool creates a solid by extruding a sketch in a linear direction. A sketch or a profile can be created on-the-fly by pressing the third mouse button in the *Selection* field. An existing sketch can be modified by selecting the Sketch icon within the *Pad Definition* window.

Open the Pad1 document, then select the Pad icon.  It has a black arrowhead beside it. Selecting the black arrowhead will display additional, related icons.

The *Pad Definition* window appears.

Select the *More* >> button. The expanded window should look like below.



Type

Dimension	Extends the pad a specified distance
Up to next	Extends the pad to the next side of a solid feature
Up to last	Extends the pad to the last face of an existing part
Up to plane	Extends the pad to a specified plane
Up to surface	Extends the pad to a specified surface
Offset	Offsets the pad from the chosen limit; only available when a *Type* other than *Dimension* is selected. A positive value extends the pad; a negative value retracts it.

Profile/Surface

Selection	Specifies the sketch to be padded; a surface can also be used
Thick	Adds a constant wall thickness to the profile

Reverse Side	Reverses the side that an open profile is padded
Mirrored extent	Extends the pad the same length in both directions; a second limit cannot be specified when this option is selected. Only available when the *Type* is *Dimension*.
Reverse Direction	Extends the pad in the opposite direction
Direction	
Normal to profile	Extends the pad normal to the sketch plane
Reference	Extends the pad in a user-specified direction; if a plane or a planar face is chosen, the extrusion direction is normal to it
Thin Pad	
Thickness1/2	Specifies the wall thickness to be applied
Neutral Fiber	Splits the *Thickness1* value in half, thereby applying the distance equally to both sides
Merge Ends	Extends or trims the pad to existing material

Select *Sketch.1*. This defines which sketch will be padded. It can be selected graphically or from the tree.

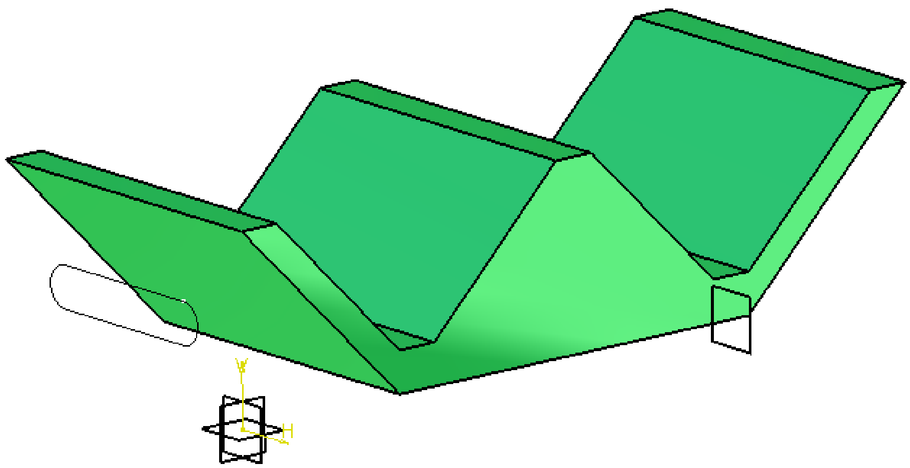
Change the *Length* value of the *First Limit* to 4.0, the select *Preview*. A solid appears in the graphical area.

Select the *Mirrored extent* checkbox option, then click *Preview*. The value is doubled so that the pad is extended 4.0 inches in both directions from the sketch plane.

Select *Mirrored extent* to turn it off, then click *Preview*. The pad extends four inches in only one direction again.

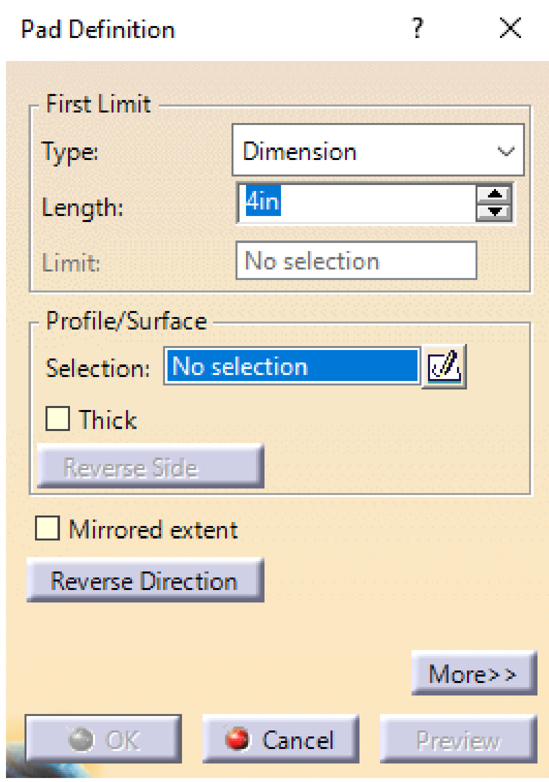
Select the *Reverse Direction* button and click *Preview*. The pad extends in the opposite direction.

Select *OK*. It is created similar to below.



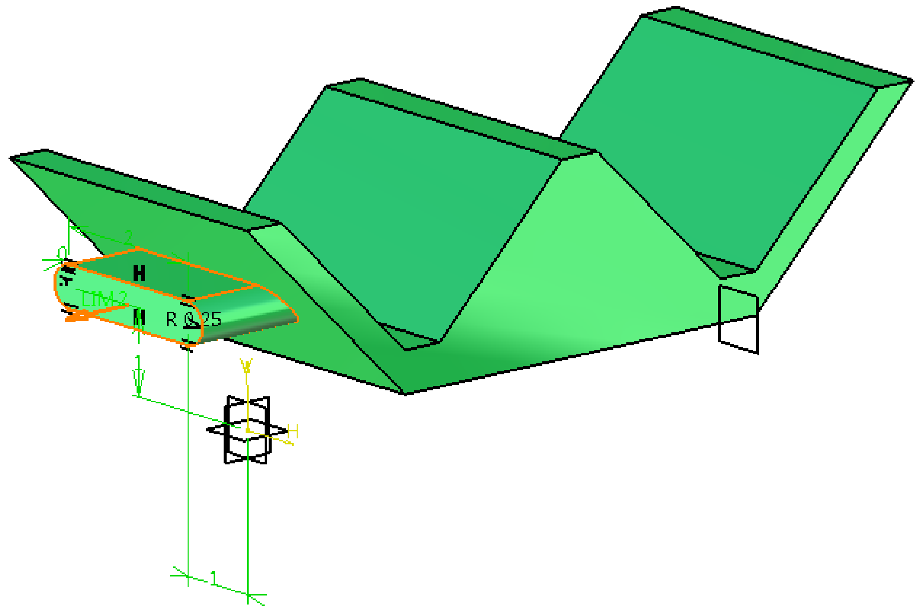
The sketch was automatically hidden. This will be the case for most sketch-based features due to a default setting in CATIA.

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

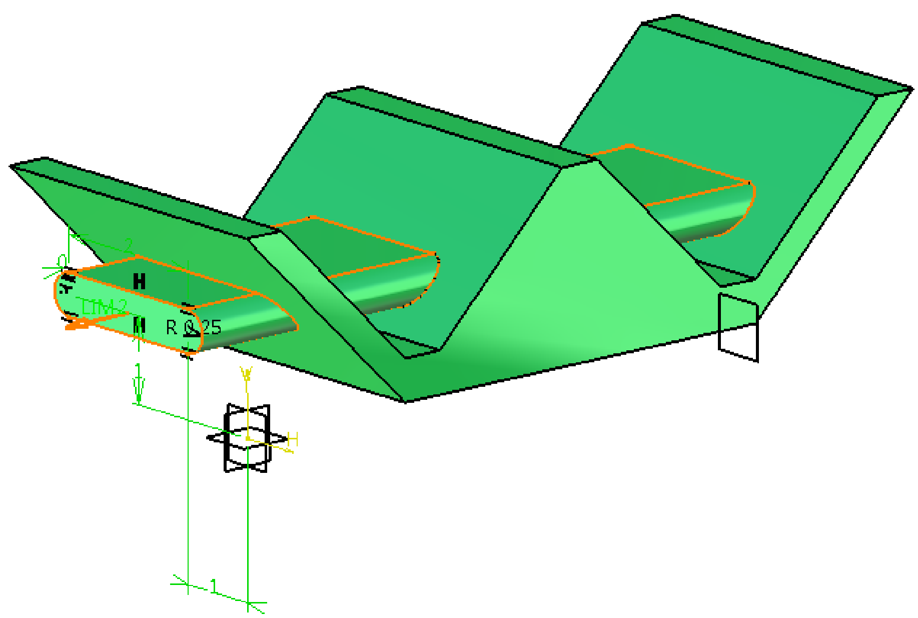


Select *Sketch.2*. This specifies the sketch to be padded.

Change the *Type* drop-down menu to *Up to next* and select *Preview*. The new pad stops at the inside wall of the first pad. It should appear similar to below.

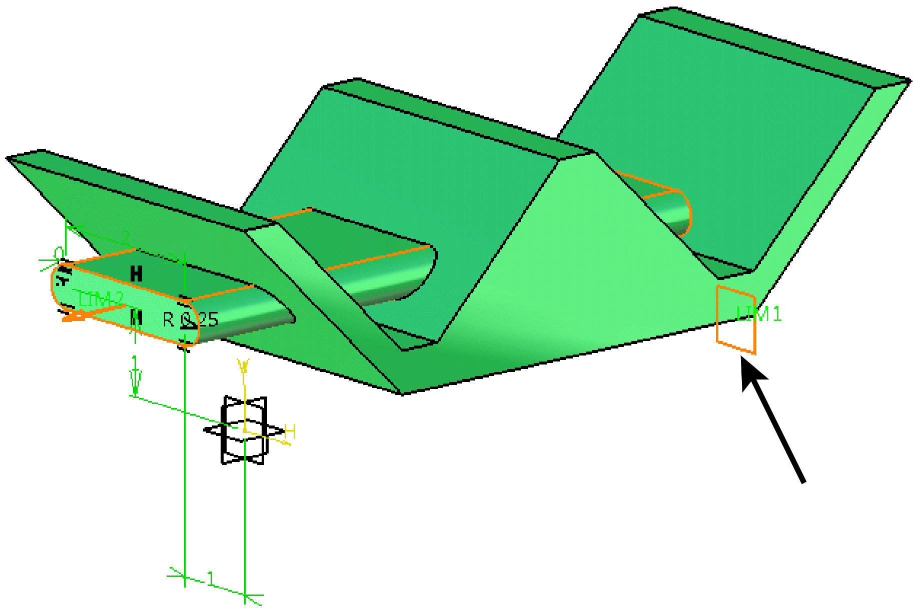


Change the *Type* drop-down menu to *Up to last* and select *Preview*. The pad extends through the entire part now.



Change the *Type* drop-down menu to *Up to plane*. A plane must be specified in order to limit the pad.

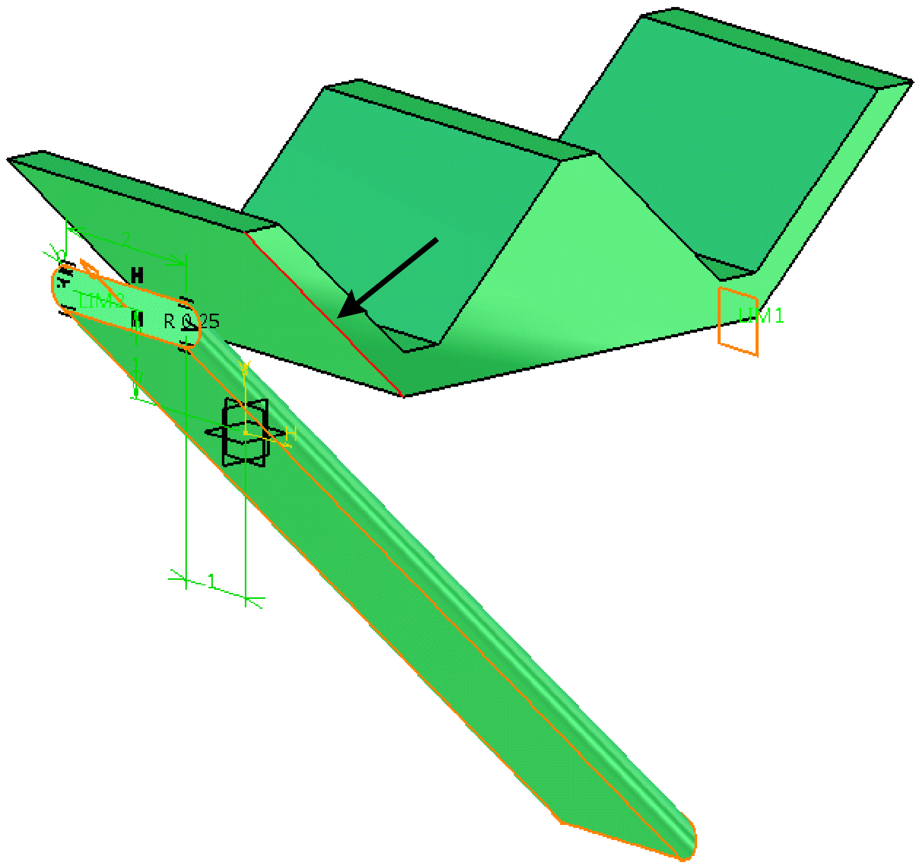
Select *Plane.1* from the geometrical set in the tree and click *Preview*. The pad stops at the plane.



The *Up to surface* option works very similar, except that the pad is limited by a surface rather than a plane. A face can act as a surface, as well.

Select the *More* >> button. The window expands.

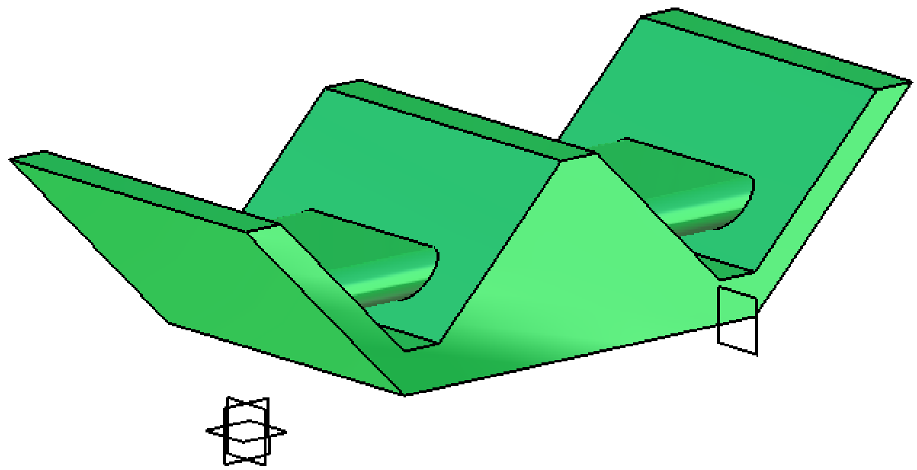
Turn off the *Normal to profile* checkbox option and select in the *Reference* field, then choose the angled edge shown below and click *Preview*. This forces the pad to extend parallel to the line. It still stops at the plane.



Select *Normal to profile*. The pad extends normal to the sketch plane again.

Select in the *Limit* field for the *First Limit*, then select the angled face closest to the sketch. A planar face can be used with the *Up to plane* option. The pad stops at the selected face.

Change the *Type* drop-down menu for the *Second Limit* to *Up to plane*, then select the angled face furthest from the sketch and click *OK*. The pad extends all the way through the part. It should appear like below.

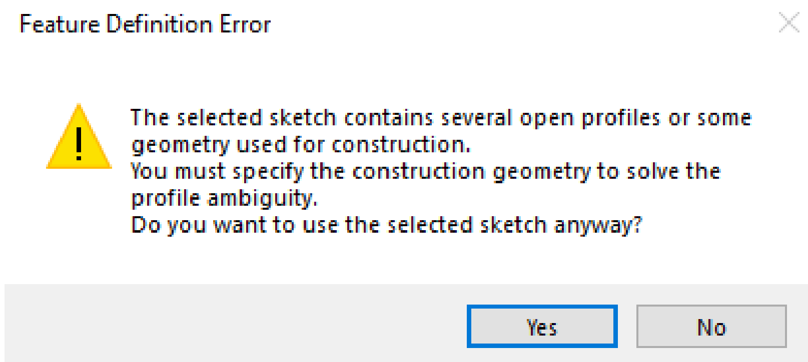


Open profiles in sketches can be used to create pads or pockets, as long as they are closed by other faces in the existing part.

Save and close the document.

Open the Pad2 document. This exercise will demonstrate the *Thin Pad* options.

Select the Pad icon, then select *Sketch.1*.  The *Pad Definition* window appears, along with the *Feature Definition Error* window. The error is triggered because the sketch contains open profiles. This will not be a problem as long as the *Thick* option is used, or as long as the open profile is limited by the existing solid.

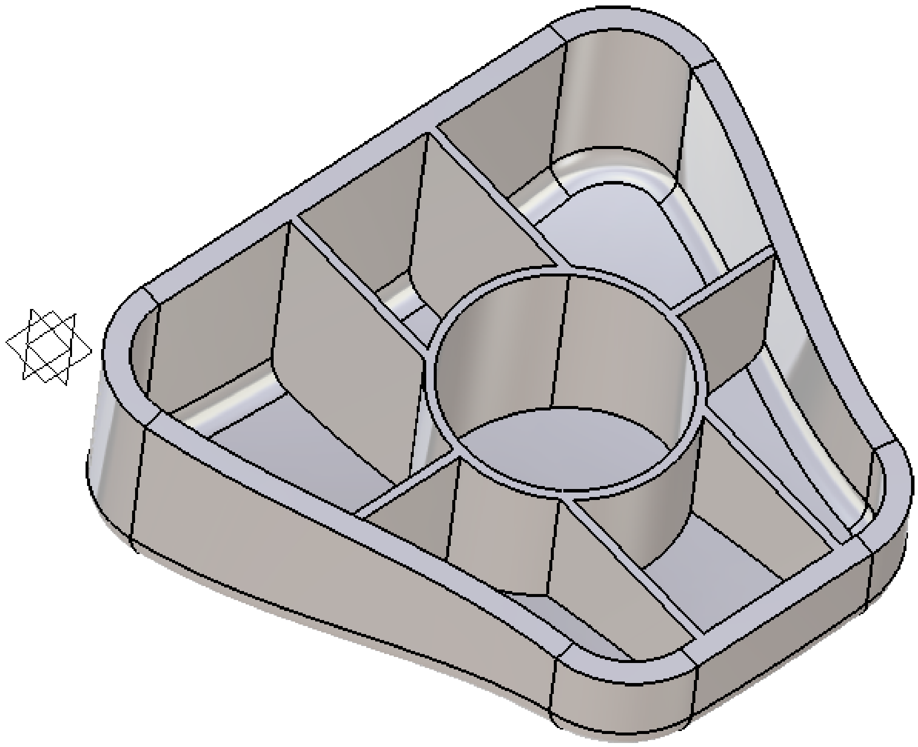


Select *Yes* to the *Feature Definition Error* window, then turn on the *Thick* option. The *Thin Pad* section of the *Pad Definition* window become available.

Turn on the *Neutral Fiber* option and enter 0.1 for *Thickness1*. This specifies that the pad will have a constant wall thickness of 0.1 inches, split evenly on either side of the sketch plane.

Reverse the direction of the pad so that it extends downward, then change the *First Limit* to *Up to surface* and select the bottom, outside surface of the part. The pad will match the contour of the selected face.

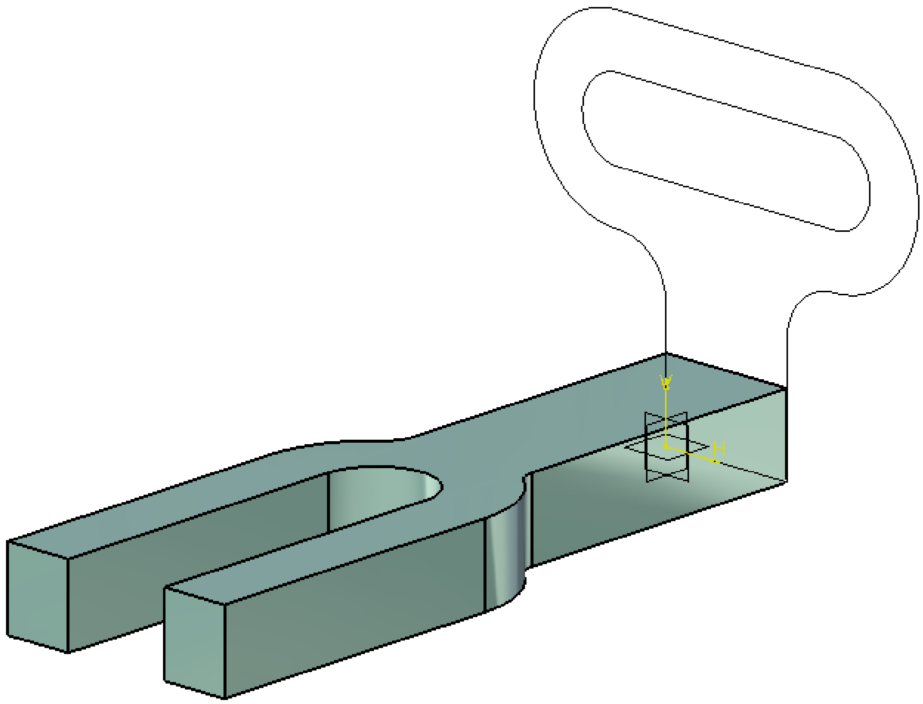
Select *OK.* It should look like below.



Save and close the document.

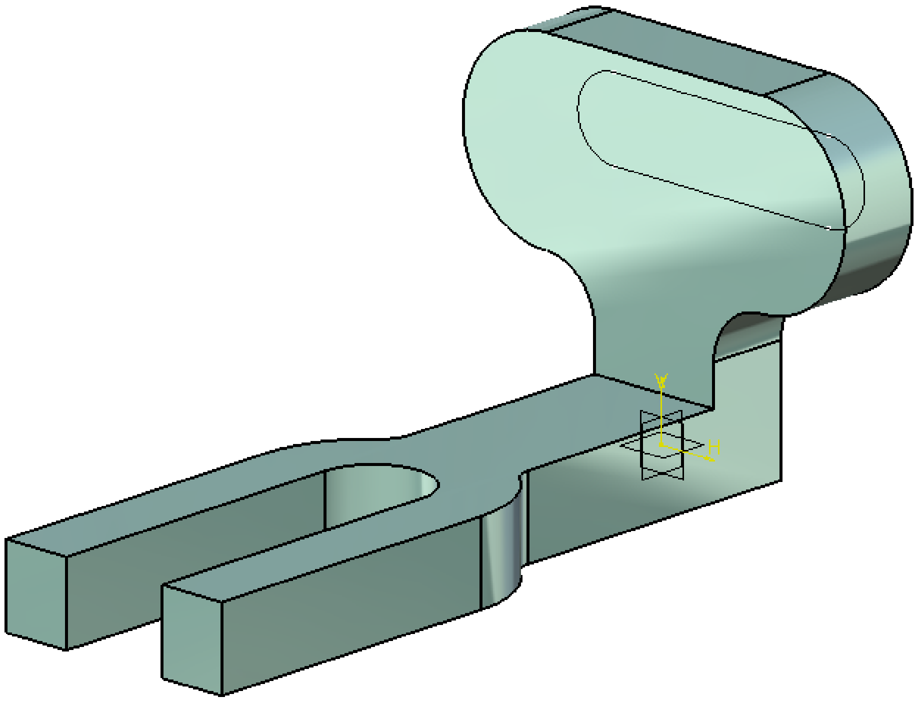
Open the Pad3 document and select the Pad icon, then select *Sketch.1*. 

Ensure the *Type* is *Dimension*, then change the *Length* to 0.75 and select *OK*. The pad should appear similar to below.



Select the Pad icon and pick *Sketch.2*. 

Ensure the *Type* is *Dimension*, then change the *Length* to 0.75 and select *OK.* When two, solid features within the same body intersect, they are automatically merged together.



Keep this document open for the next exercise.

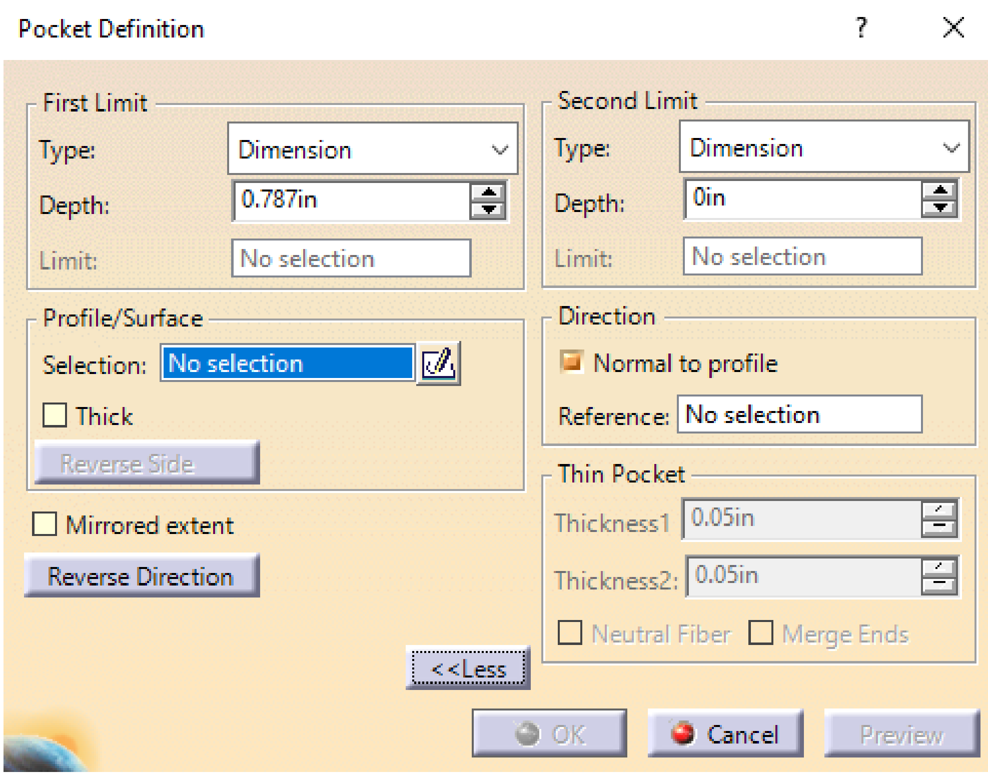
Pocket

The Pocket tool removes material from a solid by extruding a sketch in a linear direction. As with the Pad icon, a sketch or a profile can be created on-the-fly by pressing the third mouse button in the *Selection* field. An existing sketch can be modified by selecting the Sketch icon within the *Pocket Definition* window.

Select the Pocket icon.  It has a black arrowhead beside it. Selecting the black arrowhead will display additional, related icons.

The *Pocket Definition* window appears.

Select the *More* >> button. The expanded window should look like below.



The options here are exactly the same as the Pad tool. The major difference between a pocket and a pad is that a pocket is removed from the solid instead of added to it.

Type

Dimension	Extends the pocket a specified distance
Up to next	Extends the pocket to the next side of a solid feature

Up to last	Extends the pocket to the last face of an existing part
Up to plane	Extends the pocket to a specified plane
Up to surface	Extends the pocket to a specified surface
Offset	Offsets the pocket from the chosen limit; only available when a *Type* other than *Dimension* is selected. A positive value extends the pocket; a negative value retracts it.

Profile/Surface

Selection	Specifies the sketch to be pocketed; a surface can also be used
Thick	Adds a constant wall thickness to the profile
Reverse Side	Reverses the side that a profile is pocketed

Mirrored extent	Extends the pocket the same length in both directions; a second limit cannot be specified when this option is selected. Only available when the *Type* is *Dimension*.
Reverse Direction	Extends the pocket in the opposite direction

Direction

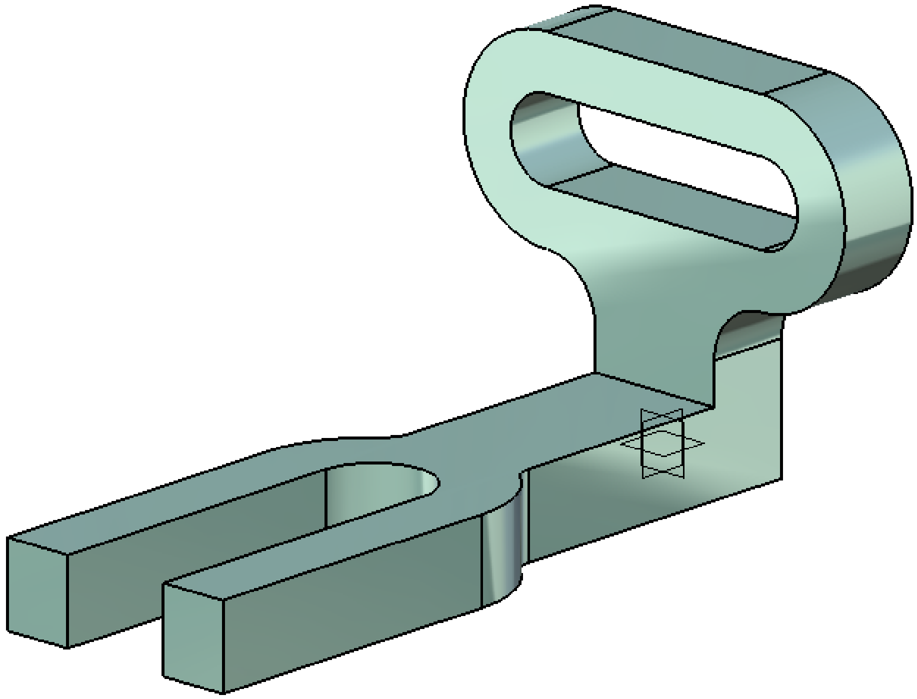
Normal to profile	Extends the pocket normal to the sketch plane
Reference	Extends the pocket in a user-specified direction; if a plane or a planar face is chosen, the extrusion direction is normal to it

Thin Pad

Thickness1/2	Specifies the wall thickness to be applied
Neutral Fiber	Splits the *Thickness1* value in half, thereby applying the distance equally to both sides
Merge Ends	Extends or trims the pocket to existing material

Select *Sketch.3*. This specifies which sketch will be pocketed.

Change the *Type* drop-down menu to *Up to next* and select *OK*. The direction of the pocket automatically reversed so that it intersects the existing solid. It should appear similar to below.



By using *Up to next* as the pocket's limit instead of a dimension, the pocket will always be linked with the length of the element it is cutting through. Thus, if the pad's thickness is increased, the pocket's length will automatically increase as well.

Save and close the document.