

TABLE OF CONTENTS

Introduction	1
Manual Format	2
Part Design & Sketcher	3
Launching 3DEXPERIENCE with My3DE app	4
Part Design Screen	6
Pull-down Menus	7
Start	7
File	8
Edit	10
View	12
Insert	18
Tools	22
Window	27
Help	28
Bottom Toolbars in Part Design	29
Right-Side Toolbars in Part Design	32
Sketcher Screen	34
Bottom Toolbars in Sketcher	35
Sketch Tools Toolbar	36
Right-Side Toolbars in Sketcher	37
Working with Documents	39
Types of documents	39
Creating a new document	40
Opening an existing document	42
Saving a document	43
Closing a document	45
Creating a new model from an existing model	46
Manipulating the Display	47
Three button mouse	47
Two button mouse	47
SpaceBall or SpaceMouse	48
Keyboard	48
Keyboard Shortcuts	49
Basic Sketcher	51
Basic Shapes	51
Creating a new part with a new sketch	52
Saving and closing the part	53
Rectangle	54
Oriented Rectangle	55
Parallelogram	56
Elongated Hole	57
Cylindrical Elongated Hole	58
Keyhole Profile	59
Polygon	60
Centered Rectangle	61

Centered Parallelogram	62
Circle	63
Three Point Circle	64
Circle Using Coordinates	65
Tri-Tangent Circle	66
Three Point Arc	67
Three Point Arc Starting With Limits	68
Arc	69
Spline	70
Connect	72
Ellipse	74
Parabola by Focus	75
Hyperbola by Focus	76
Conic	77
Line	82
Infinite Line	83
Bi-tangent Line	84
Bisecting Line	85
Line Normal to Curve	86
Axis	88
Point by Clicking	89
Point by Using Coordinates	90
Equidistant points	91
Intersection Point	93
Projection Point	94
Align Points	96
Profile	98
Constraints	116
Dimensional Constraints	116
Geometrical Constraints	116
Operations on Profiles	163
Corner	164
Tangent Arc	169
Chamfer	170
Relimitations	177
Practicing with Constraints	181
Specification Tree	183
Hide/Show	185

Basic Part Design	187
Basic Shapes	187
Pad	187
Pocket	199
Multiple Profiles	203
Multi-Pad and Multi-Pocket	206
Shaft	209
Groove	214
Hole	217
Rib	232
Slot	236
Solid Combine	238
Stiffener	240
Multi-Sections Solids	242
Removed Multi-Sections Solid	244
Operations on Shapes	245
Edge Fillet	245
Face-Face Fillet	250
Tritangent Fillet	252
Chamfer	273
Draft Angle	282
Draft Reflect Line	290
Shell	292
Thickness	295
Thread/Tap	297
Remove face	300
Replace face	303
Modifying values	305
Interfacing with Sketcher	311
 Advanced Sketcher	 317
3-D Elements on Sketch Plane	317
Construction Geometry	324
Advanced Constraints	328
Sketch Transformations	339
Mirror	339
Symmetry	341
Translate	342
Offset	345
Rotate	349
Scale	352
Sketch Analysis	354
Sketch Visualization	359

Advanced Part Design	361
Part Transformations	361
Translation	361
Rotation	364
Symmetry	365
AxisToAxis	366
Mirror	367
Scaling	368
Affinity	369
Patterns	371
Modifying Parts	407
Modifying Parameters	407
Inserting Objects	409
Scanning the Specification Tree	411
Modifying Properties	413
Replacing Sketches	419
Changing a Sketch Support	420
Positioned Sketches	422
Cut, Copy, and Paste	425
Reordering the Specification Tree	428
Inserting Bodies and Boolean Operations	435
Inserting Part Bodies	435
Boolean operations	437
Part Design Using Surfaces	447
Split	447
Sew Surface	449
Thick Surface	451
Close Surface	454
Annotations	455
Applying Materials	459
Sectioning	462
Delete Useless Elements	466
Problems	467
Problem #1.0	467
Problem #2.0	468
Problem #3.0	469
Problem #4.0	470
Problem #5.0	471
Problem #6.0	472
Problem #7.0	473
Problem #8.0	474
Problem #9.0	475
Problem #10.0	476
Problem #11.0	477
Problem #12.0	478
Problem #13.0	479
Problem #14.0	480

Problem #15.0	481
Problem #16.0	482
Problem #17.0	483
Problem #18.0	484
Problem #19.0	485
Problem #20.0	486
Problem #21.0	487
Problem #22.0	488
Problem #23.0	489
Problem #24.0	491
Problem #25.0	492
Problem #26.0	493
Problem #27.0	494
Problem #28.0	495
Problem #29.0	496
Problem #30.0	497
Problem #31.0	498
Problem #32.0	499
Problem #33.0	500
Problem #34.0	501
Problem #35.0	502
Appendix A	503
Customize - Start Menu	503
Customize - User Workbenches	505
Customize - Toolbars	506
Customize - Commands	507
Customize - Options	508
Appendix B	509
General - PCS	509
General - Display - Tree Appearance	510
General - Display - Tree Manipulation	511
General - Display - Visualization	512
General - Parameters and Measure - Units	514
General - Parameters and Measure - Constraints and Dimensions	515
Infrastructure - Product Structure - Product Structure	516
Infrastructure - Part Infrastructure - General	517
Infrastructure - Part Infrastructure - Display	518
Infrastructure - Part Infrastructure - Part Document	519
Mechanical Design - Part Design	520
Mechanical Design - Sketcher	521

Appendix C	523
Material Library	523
Construction	523
Fabrics	524
Metal	525
Other	526
Painting	527
Shape Review	528
Stone	529
Wood	530
List mode	531
Applying a material	532
Properties of a material	533
Rendering - Lighting tab	533
Rendering - Texture tab	534
Inheritance	535
Feature Properties	536
Analysis	537
Composites	538
Drawing	539
Appendix D	541
Reference Geometry	541
Offset from plane	541
Parallel through point	543
Angle/Normal to plane	544
Through three points	545
Through two lines	546
Through point and line	547
Through planar curve	548
Normal to curve	549
Equation	550
Tangent to surface	551
Mean through points	551
Appendix E	553
Measurement Tools	553
Measure Between	553
Measure Item	561
Measure Inertia	566
Appendix F	569
Advanced Dress-Up Features	569
Draft Both Sides	569
Advanced Draft	579
Automatic Draft	582
Automatic Filletting	584

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



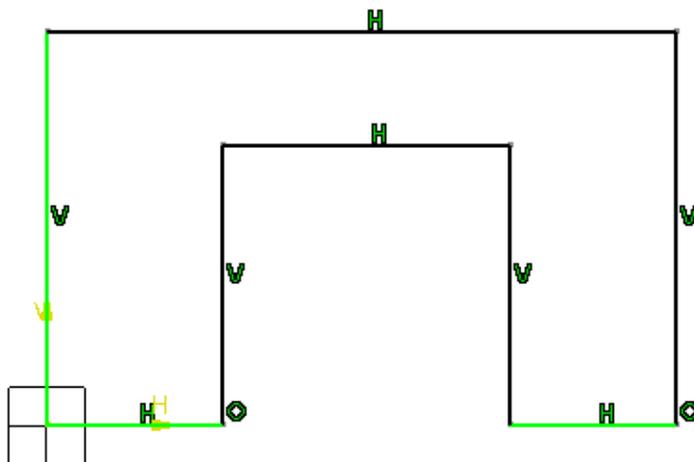
Three Point Arc

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.

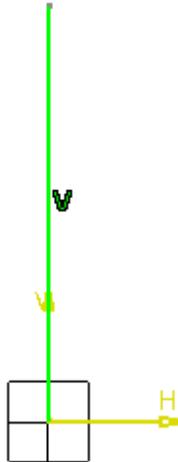


Select the **Profile** icon.

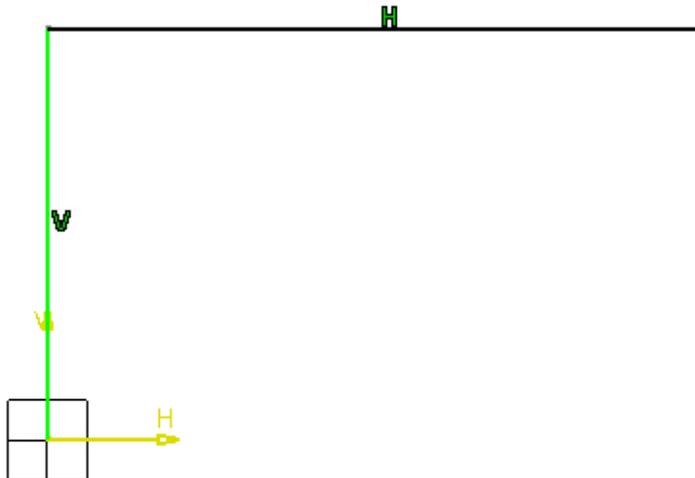


Select the **origin point of the sketch plane**. This defines the start point of the profile.

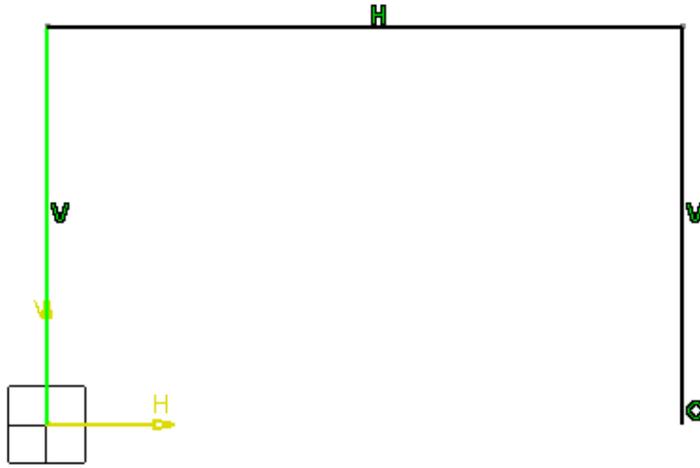
Select a location straight above the origin. Ensure the line is blue before selecting the next point. Doing so will create a vertical constraint.



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.

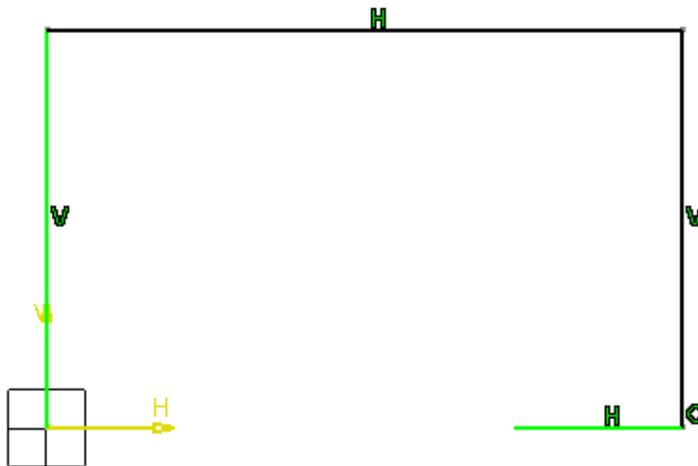


Select straight below the previous location and on the *H* axis. A vertical constraint and a coincidence constraint are created. The coincidence constraint is between the end point of the new line and the sketch's *H* axis.

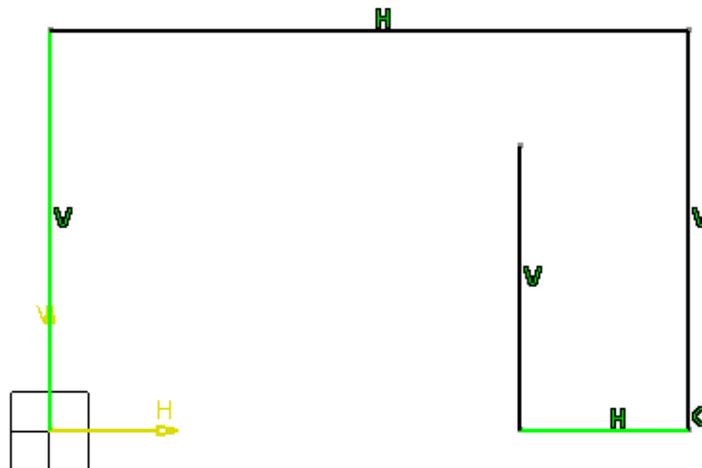


Note: At any point during the profile's creation, selecting the Undo icon or Ctrl+Z will undo the previous selection. You can undo as many selections in a row as necessary.

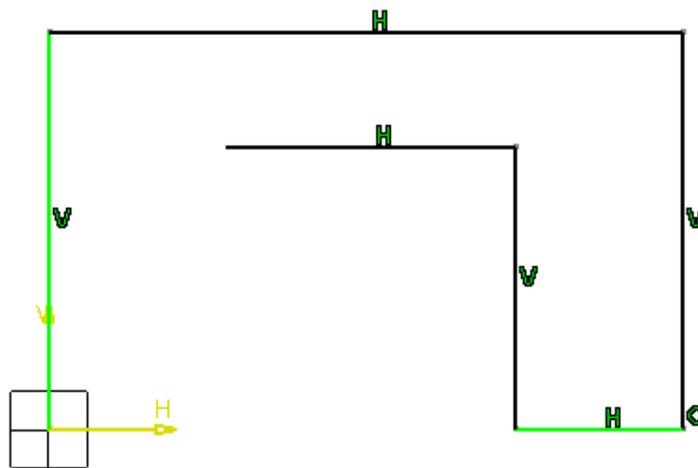
Select a location straight to the left of the previous end point. A horizontal constraint is created, and the line turns green. When an element is fully green, it is iso-constrained and cannot be moved. This will be discussed more later.



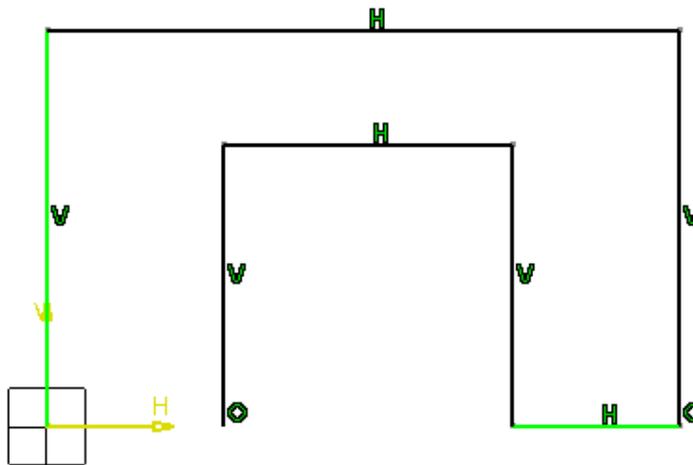
Select a location straight above the previous point. It should appear similar to below.



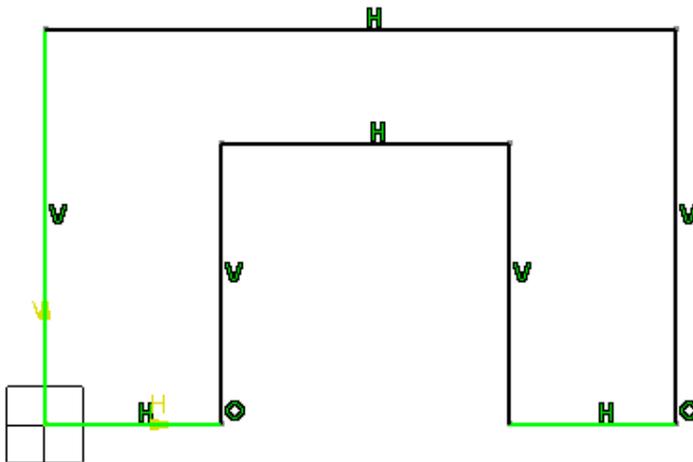
Select a location straight to the left of the previous location.



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

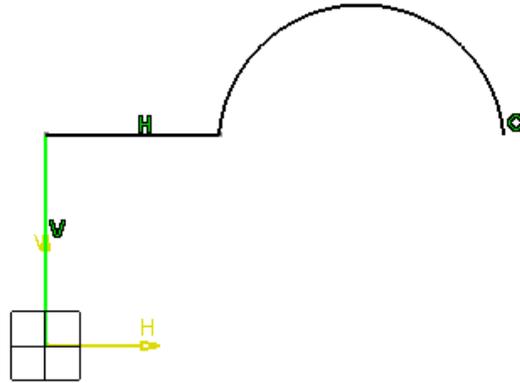


Select the sketch origin again for the last point. By selecting the same location for the start and end points in a single operation, the Profile command is automatically exited. The sketch should appear similar to below.

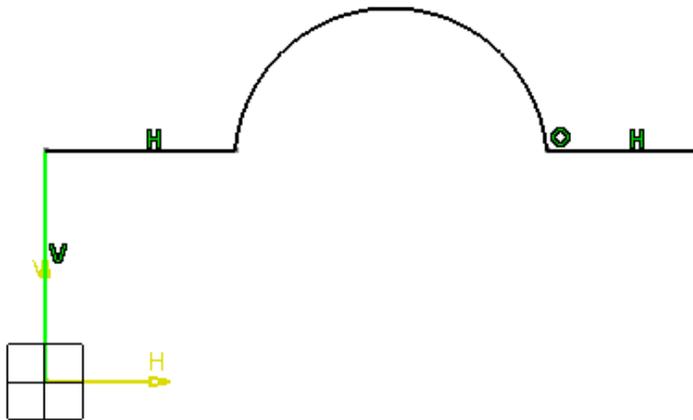


Save the document as Profile1 and close it.

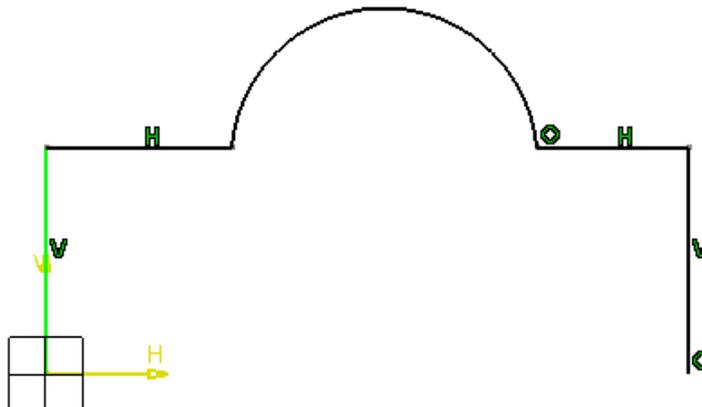
Move the mouse cursor down and to the right of the previous location, then click in space when you see a thin, blue, solid line appear. This location should be straight across from the start of the arc. Once the three point arc is complete, the **Line** icon in the Sketch tools toolbar is automatically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



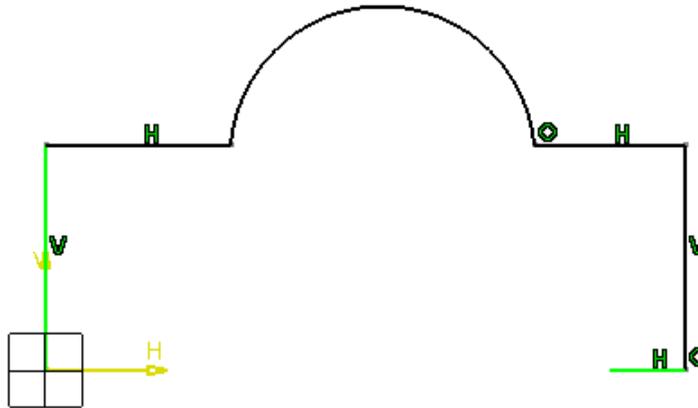
Select straight to the right of the previous location.



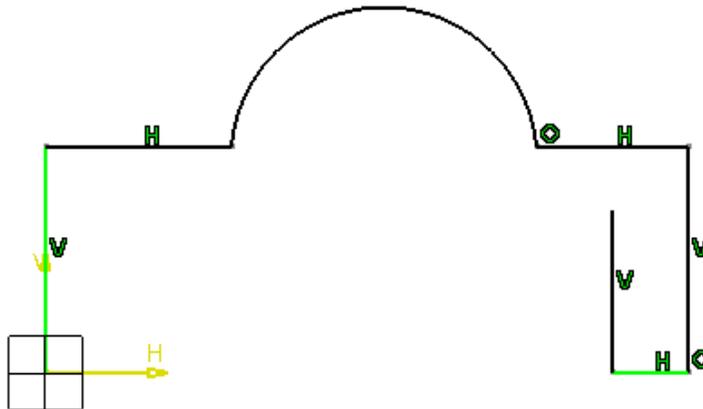
Select straight below the previous point and on the H axis.



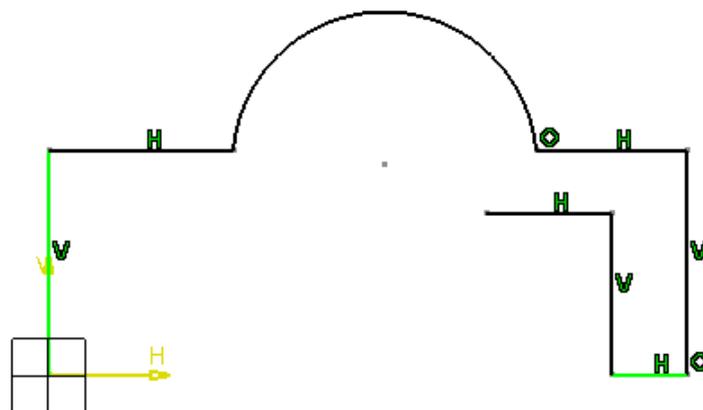
Select straight to the left of the previous location.



Select straight above the previous location.



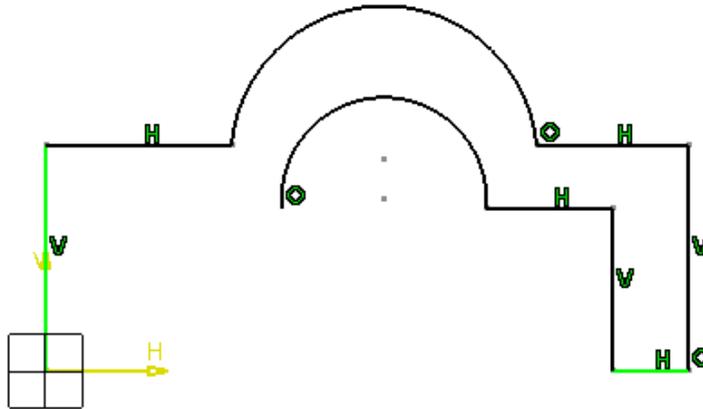
Select straight to the left of the previous location. It should appear similar to below.



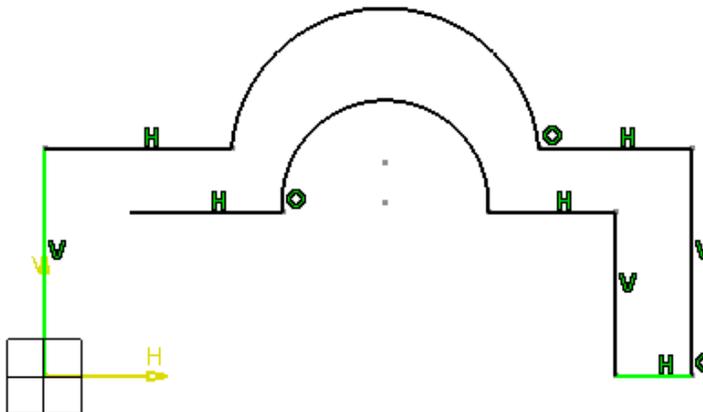
Select the **Three Point Arc** icon in the Sketch tools toolbar.  The arc will begin at the last profile location specified.

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

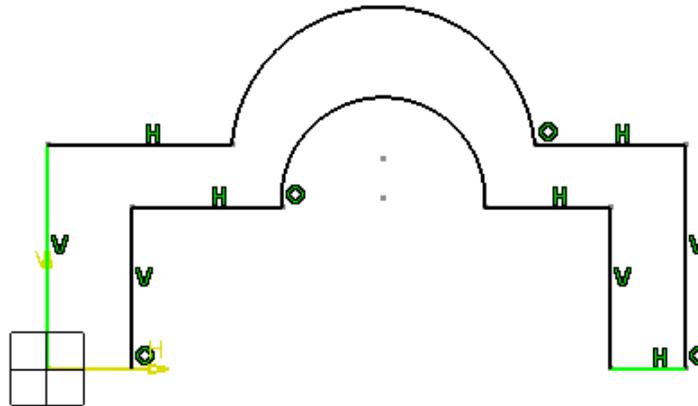
Move the mouse cursor down and to the left of the previous location, then click in space when you see a thin, blue, solid line appear. This location should be straight across from the start of the arc. Once the three point arc is complete, the **LINE** icon in the Sketch tools toolbar is automatically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



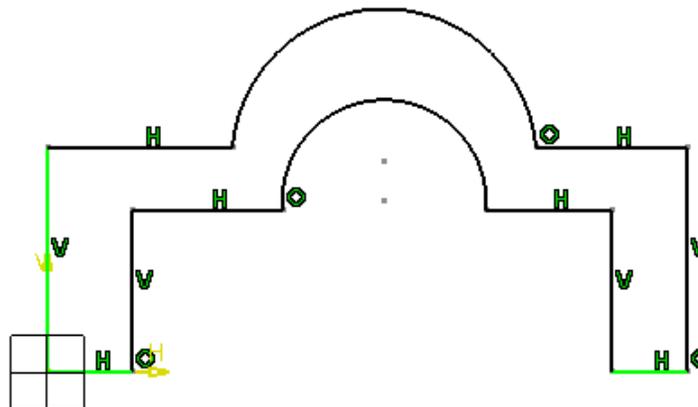
Select straight to the left of the previous location.



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 Profile2 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

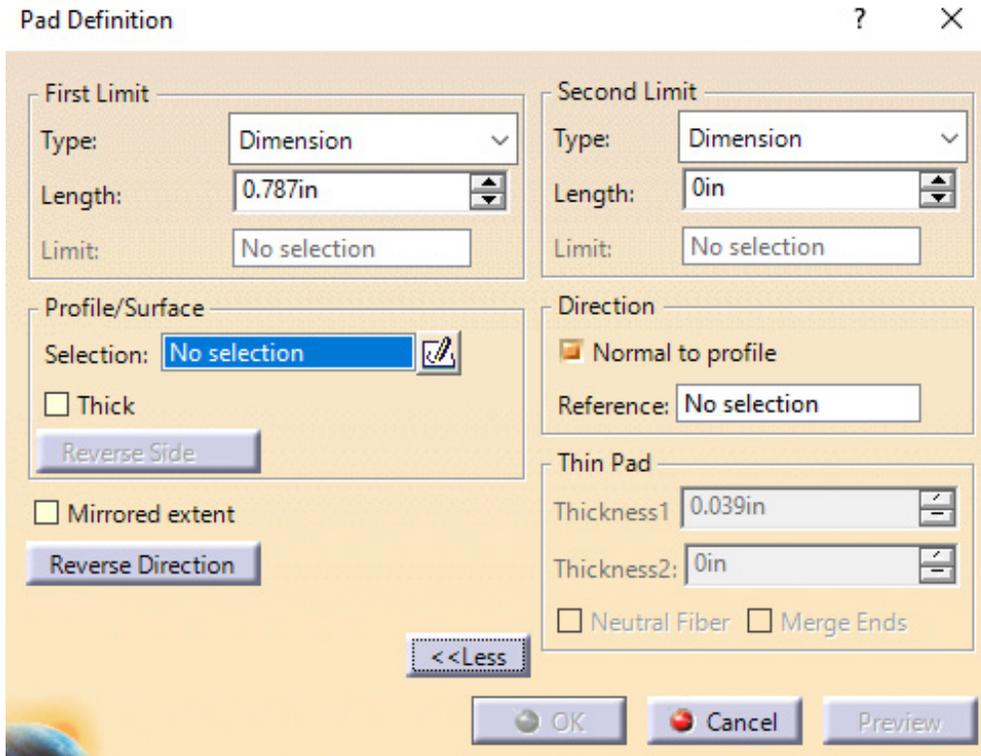
The **Pad** tool creates a solid by extruding a sketch in a linear direction. A sketch or a profile can be created on-the-fly by pressing the third mouse button in the *Selection* field. An existing sketch can be modified by selecting the **Sketch** icon within the *Pad Definition* window.

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

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

Profile/Surface

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

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

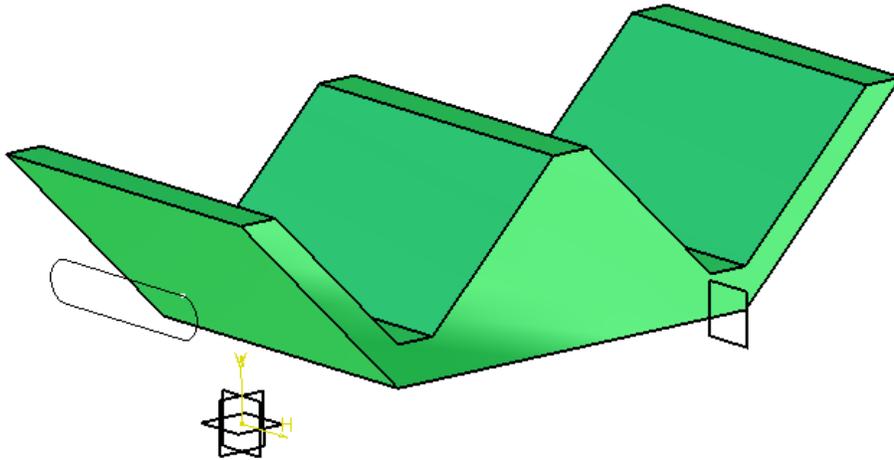
Change the *Length* value of the *First Limit* to 4.0, the select *Preview*. A solid appears in the graphical area.

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.

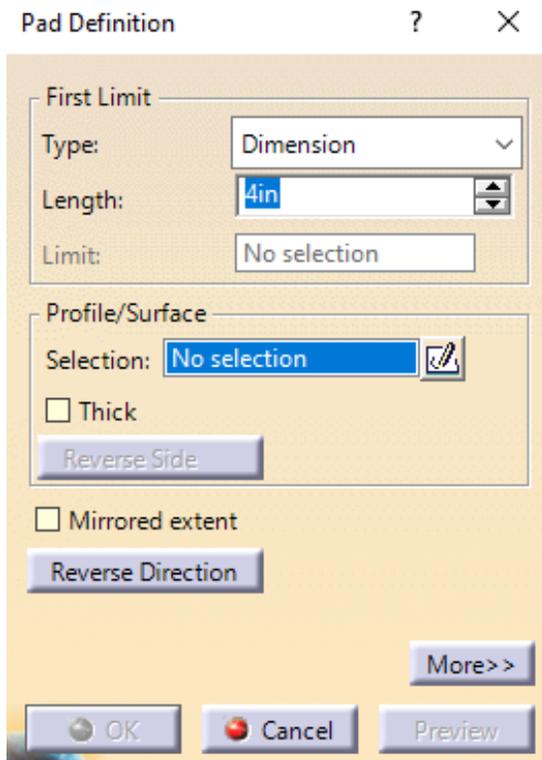
Select the *Reverse Direction* button and click *Preview*. The pad extends in the opposite direction.

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



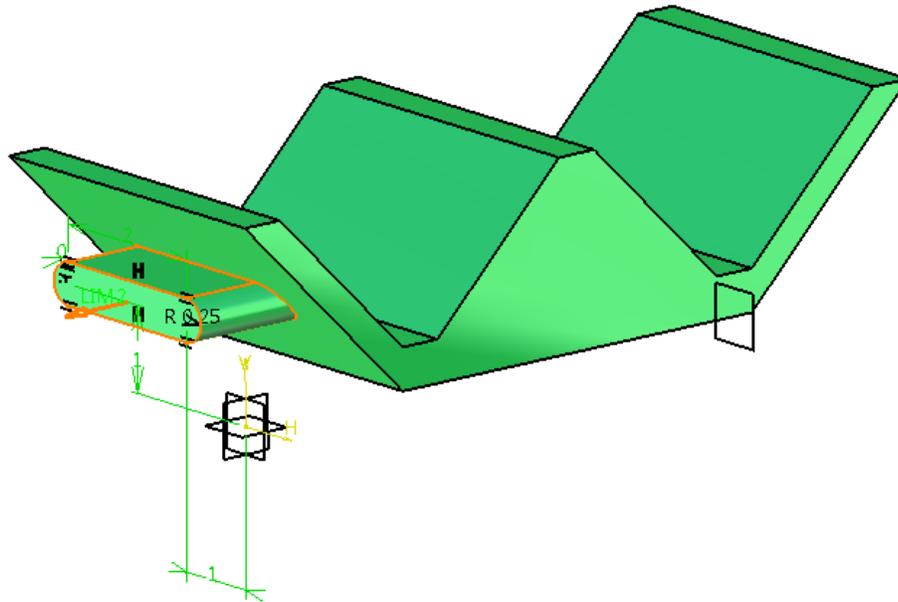
The sketch was automatically hidden. This will be the case for most sketch-based features due to a default setting in CATIA.

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

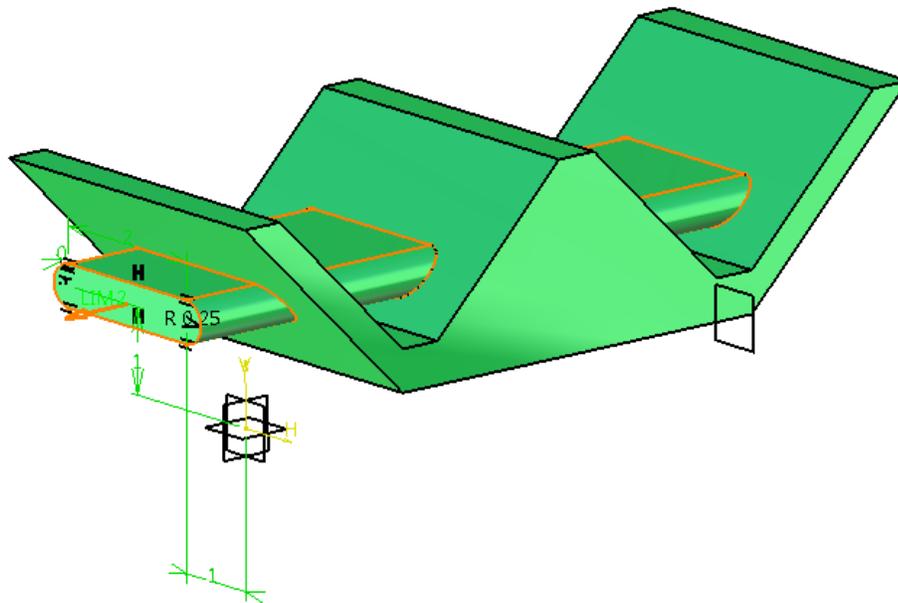


Select **Sketch.2**. This specifies the sketch to be padded.

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.

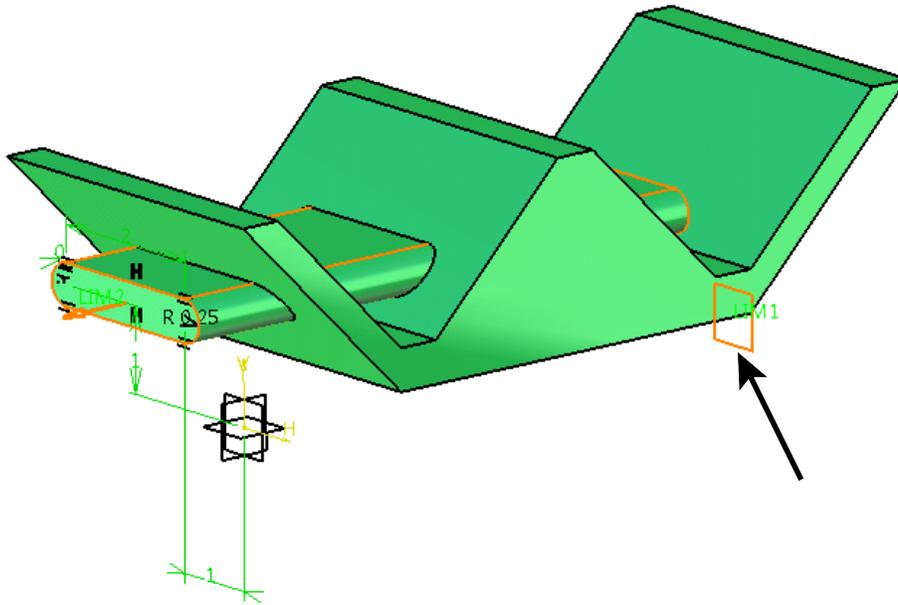


Change the *Type* drop-down menu to *Up to last* and select *Preview*. The pad extends through the entire part now.



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

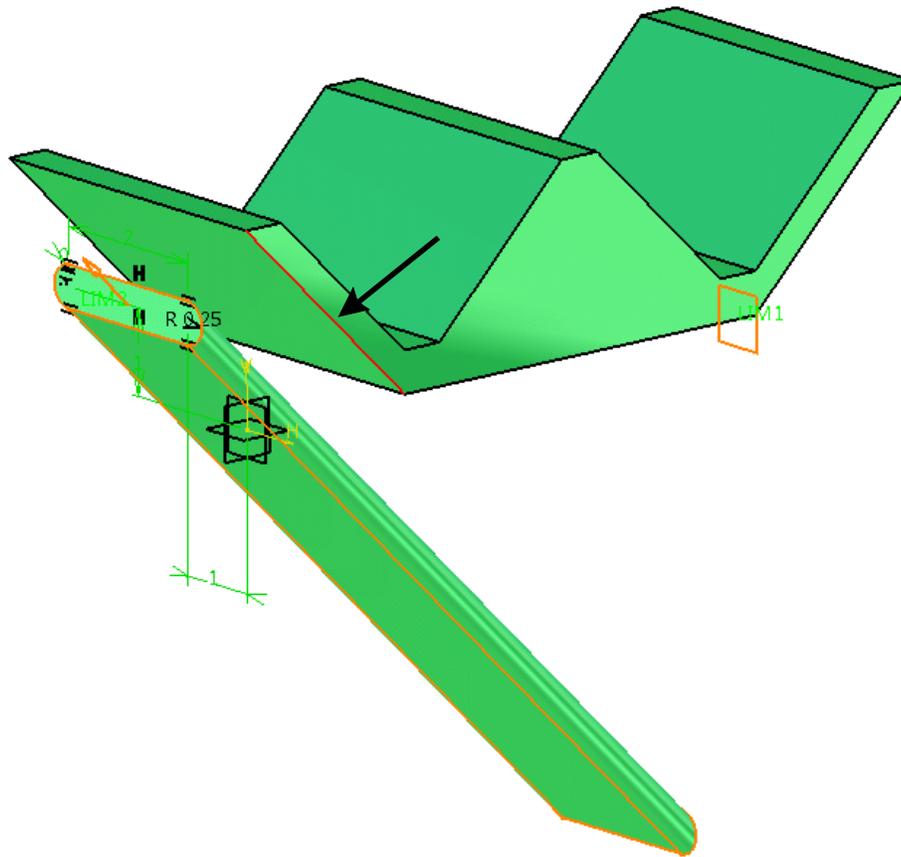
Select *Plane.1* from the geometrical set in the tree and click *Preview*. The pad stops at the plane.



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.

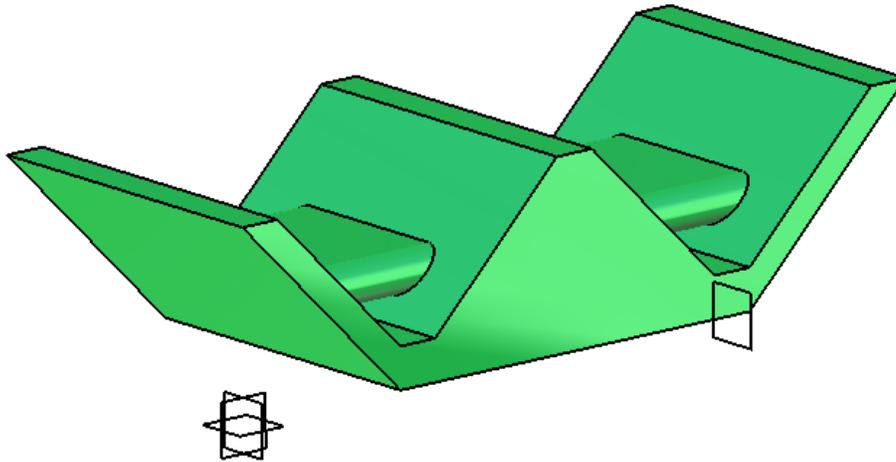
Turn off the *Normal to profile* checkbox option and select in the *Reference* field, then choose the angled edge shown below and click *Preview*. This forces the pad to extend parallel to the line. It still stops at the plane.



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.

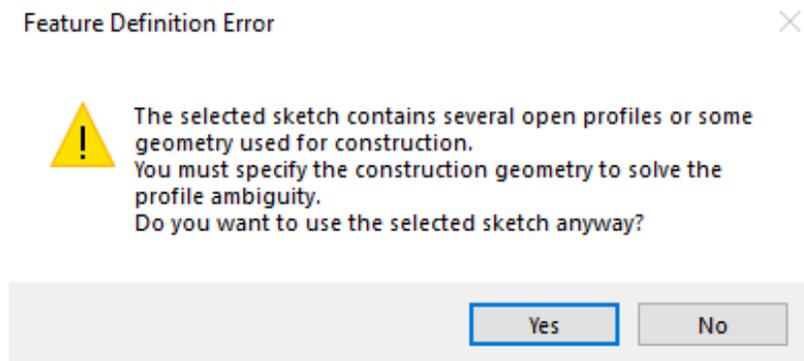


Open profiles in sketches can be used to create pads or pockets, as long as they are closed by other faces in the existing part.

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.

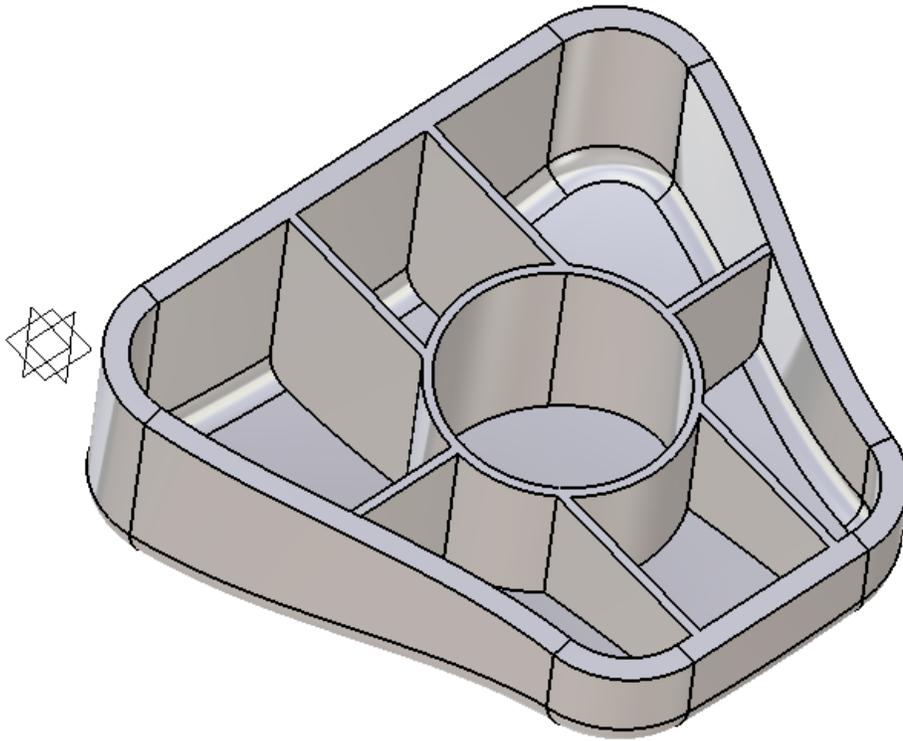


Select **Yes** to the *Feature Definition Error* window, then turn on the *Thick* option. The *Thin Pad* section of the *Pad Definition* window become available.

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.

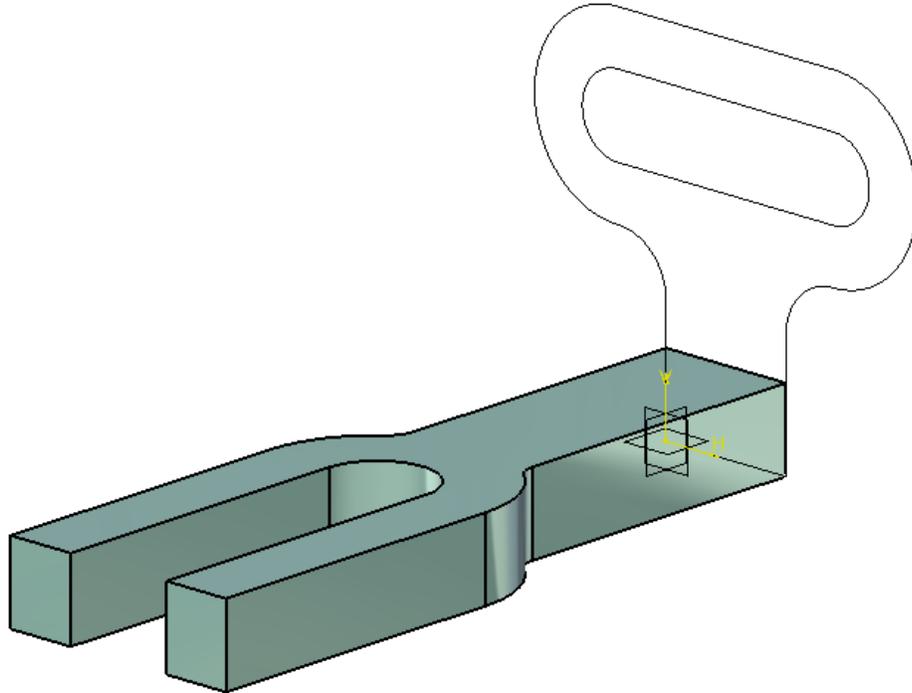
Select **OK**. It should look like below.



Save and close the document.

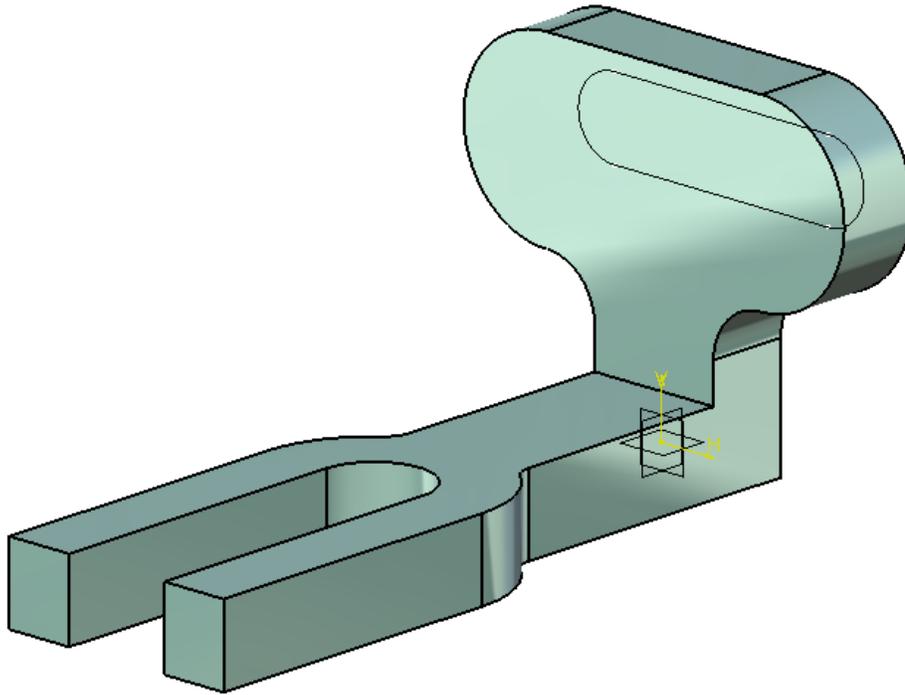
Open the **Pad3** document and select the **Pad** icon, then select *Sketch.1*. 

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

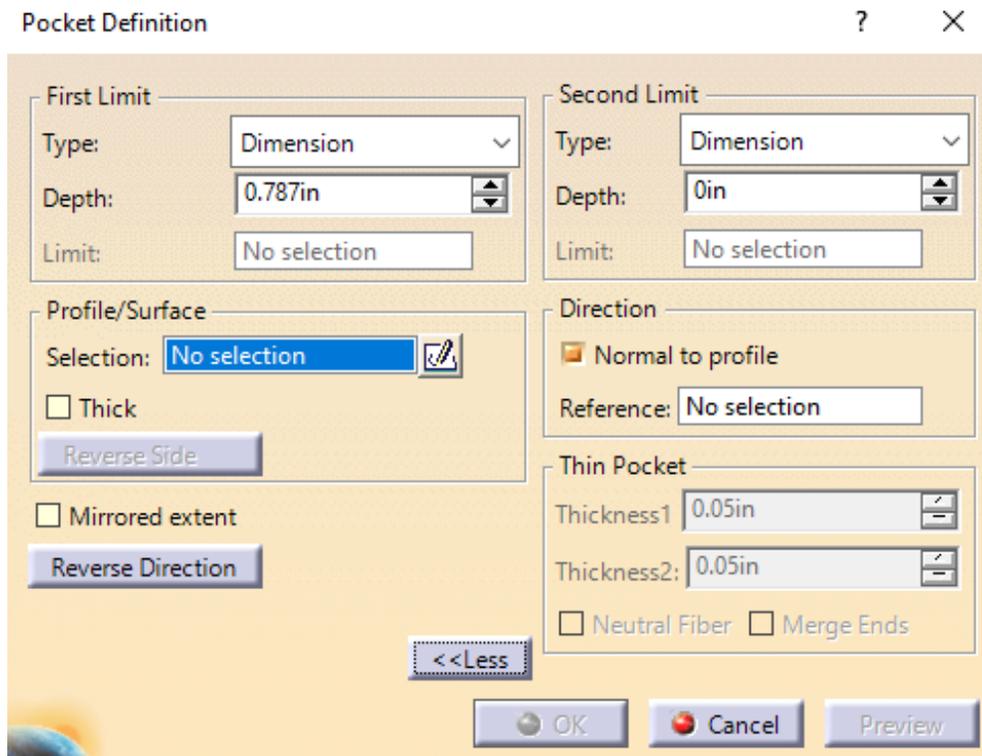
The **Pocket** tool removes material from a solid by extruding a sketch in a linear direction. As with the **Pad** icon, a sketch or a profile can be created on-the-fly by pressing the third mouse button in the *Selection* field. An existing sketch can be modified by selecting the **Sketch** icon within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

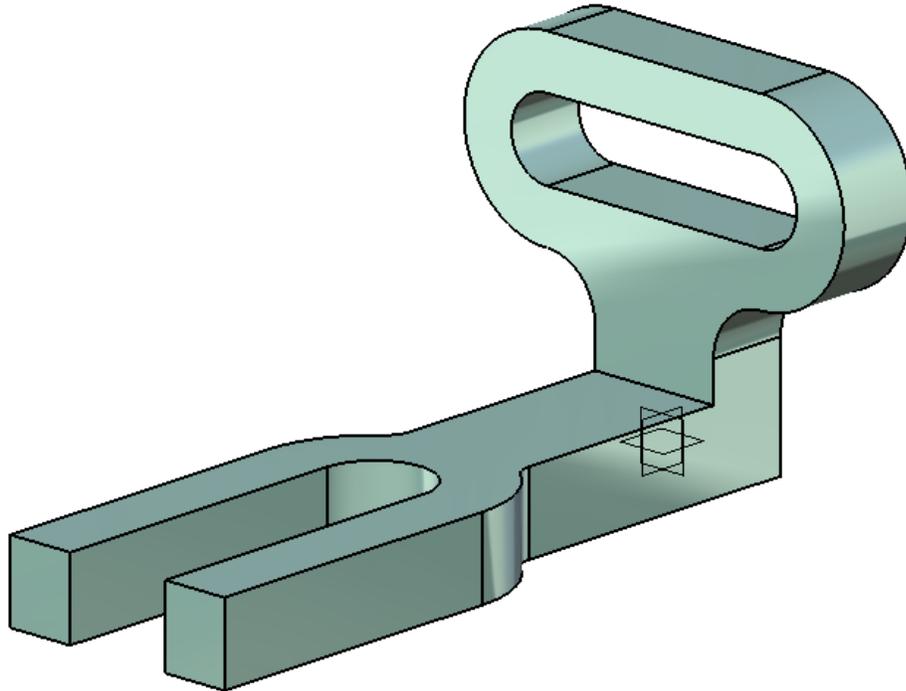
Type

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

<i>Up to last</i>	Extends the pocket to the last face of an existing part
<i>Up to plane</i>	Extends the pocket to a specified plane
<i>Up to surface</i>	Extends the pocket to a specified surface
<i>Offset</i>	Offsets the pocket from the chosen limit; only available when a <i>Type</i> other than <i>Dimension</i> is selected. A positive value extends the pocket; a negative value retracts it.
<i>Profile/Surface</i>	
<i>Selection</i>	Specifies the sketch to be pocketed; a surface can also be used
<i>Thick</i>	Adds a constant wall thickness to the profile
<i>Reverse Side</i>	Reverses the side that a profile is pocketed
<i>Mirrored extent</i>	Extends the pocket the same length in both directions; a second limit cannot be specified when this option is selected. Only available when the <i>Type</i> is <i>Dimension</i> .
<i>Reverse Direction</i>	Extends the pocket in the opposite direction
<i>Direction</i>	
<i>Normal to profile</i>	Extends the pocket normal to the sketch plane
<i>Reference</i>	Extends the pocket in a user-specified direction; if a plane or a planar face is chosen, the extrusion direction is normal to it
<i>Thin Pad</i>	
<i>Thickness1/2</i>	Specifies the wall thickness to be applied
<i>Neutral Fiber</i>	Splits the <i>Thickness1</i> value in half, thereby applying the distance equally to both sides
<i>Merge Ends</i>	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.