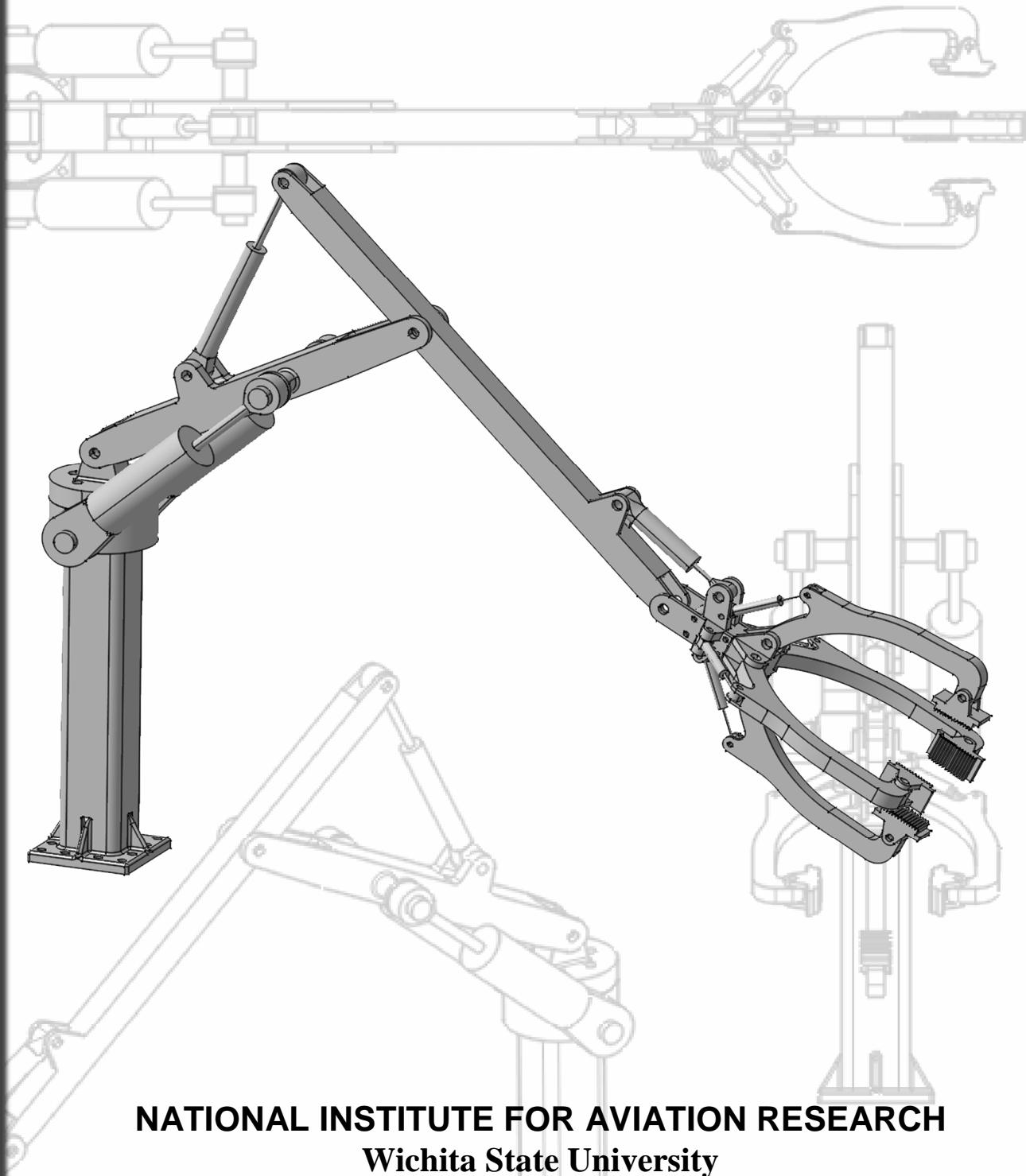


CATAV5

Part Design & Sketcher



**NATIONAL INSTITUTE FOR AVIATION RESEARCH
Wichita State University**

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Basic Part Design

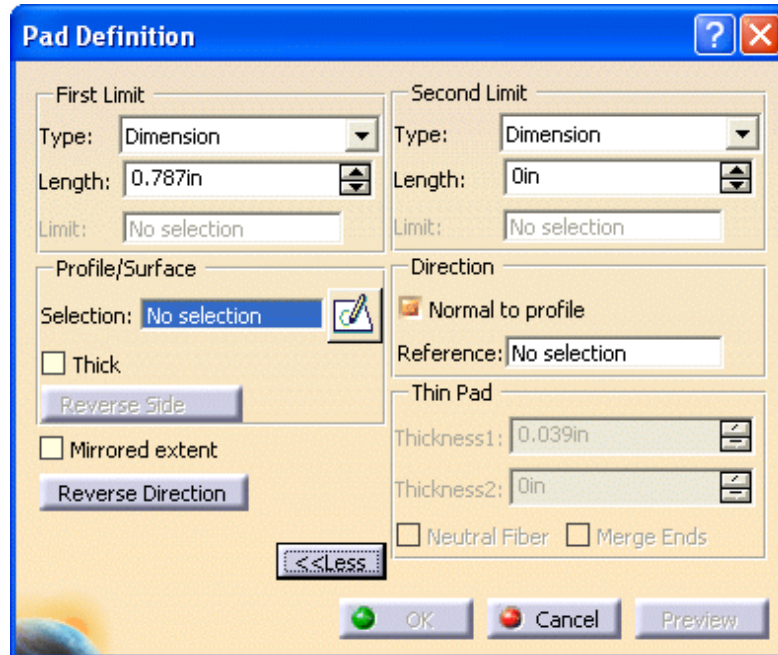
This section will cover the basic use of the Part Design workbench to create parts. This section will consist of three parts: basic shapes, operations on shapes and interfacing between part design and sketcher.

Basic Shapes

This part will discuss the various shapes that can be created in part design using the icons on the Part Design workbench. The purpose of this group of exercises is to introduce how to use those icons and their options. The usefulness of them, depend on the part you are trying to create. It is important for you to understand how to use each of these icons in conjunction with your sketches to produce your final part.

Pad

The pad icon allows you to use a sketch and extrude it in a linear direction producing a solid pad. You can create a sketch or profile on-the-fly by pressing the third mouse button while in the *Selection* box. When you create a pad, a *Pad Definition* window appears like the one shown below.



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

Type


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

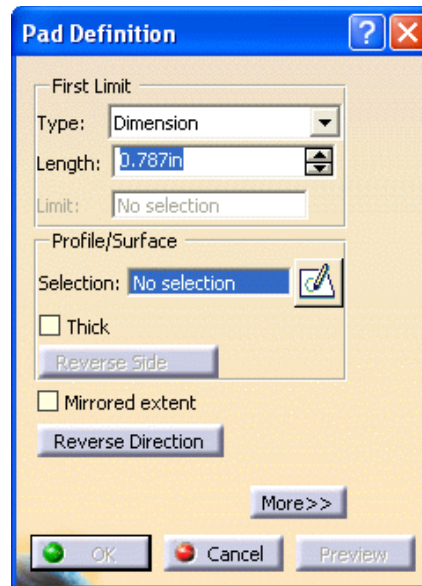
When you select a *Type* other than *Dimension* you will have the option to specify an *Offset* value from the corresponding limit.

Profile/Surface

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

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

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



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

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

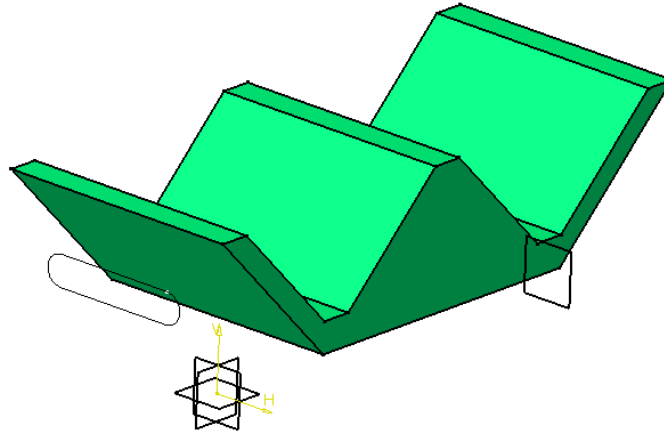
Select **Preview**. A preview of what the pad will look like appears. You will now change some of the other options to see the difference between them.

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

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

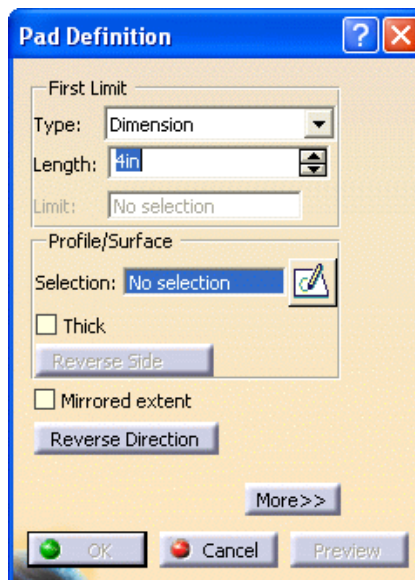
Select **Reverse Direction** and select **Preview**. Notice that the pad is still going to be four inches wide but it is now going in the opposite direction. This is the pad you want to create.

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



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

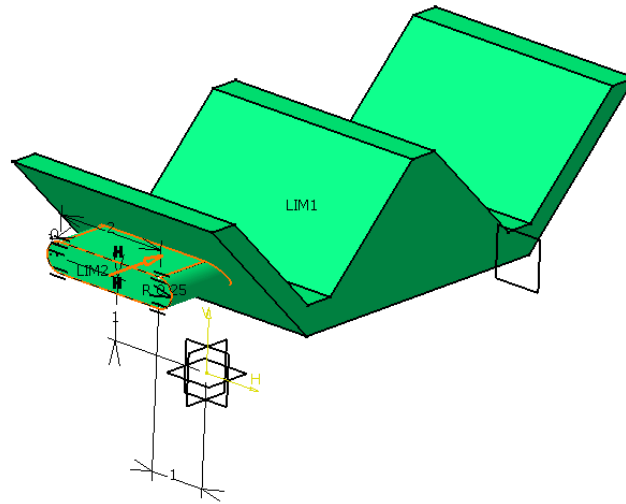
Select the pad icon.  A *Pad Definition* window appears as shown below.



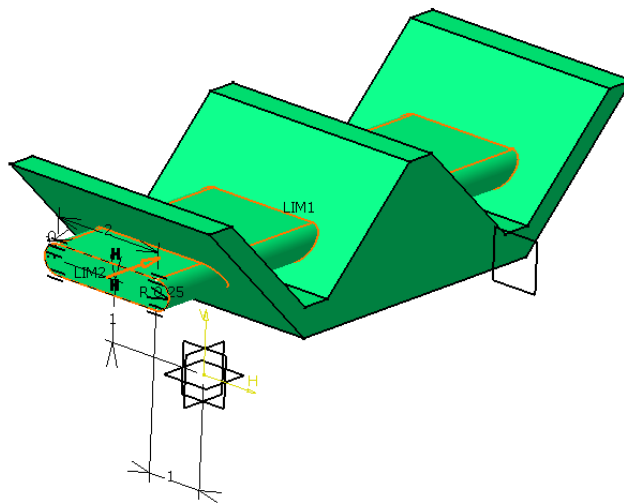
Select Sketch.2. This specifies the sketch that you want to use to create the next pad.

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

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

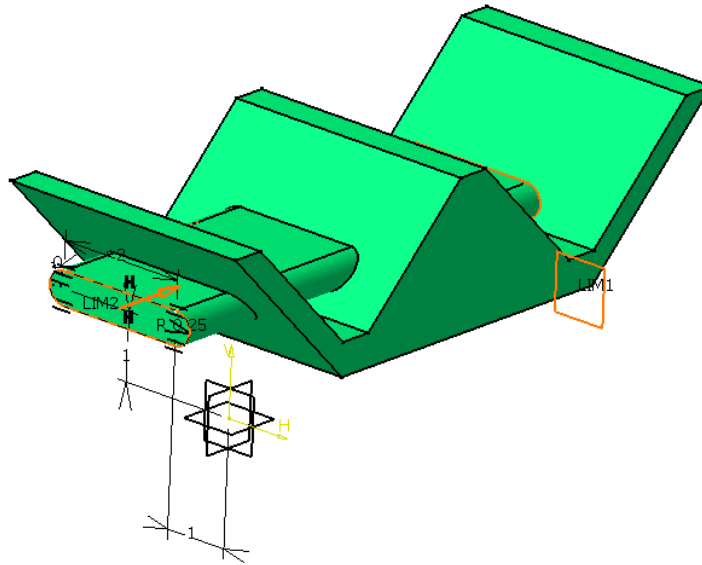


Change the *Type* to *Up to last* and select *Preview*. Notice that the pad goes all the way to the last side of the previous part. It should appear similar to the diagram shown below.



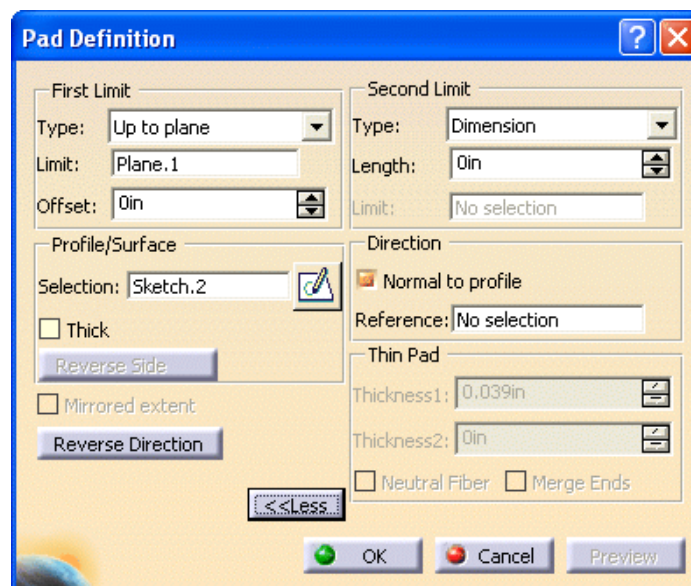
Change the *Type* to *Up to plane*. When you use this option you have to specify a plane or a planar side that you want the pad to be limited by.

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



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

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

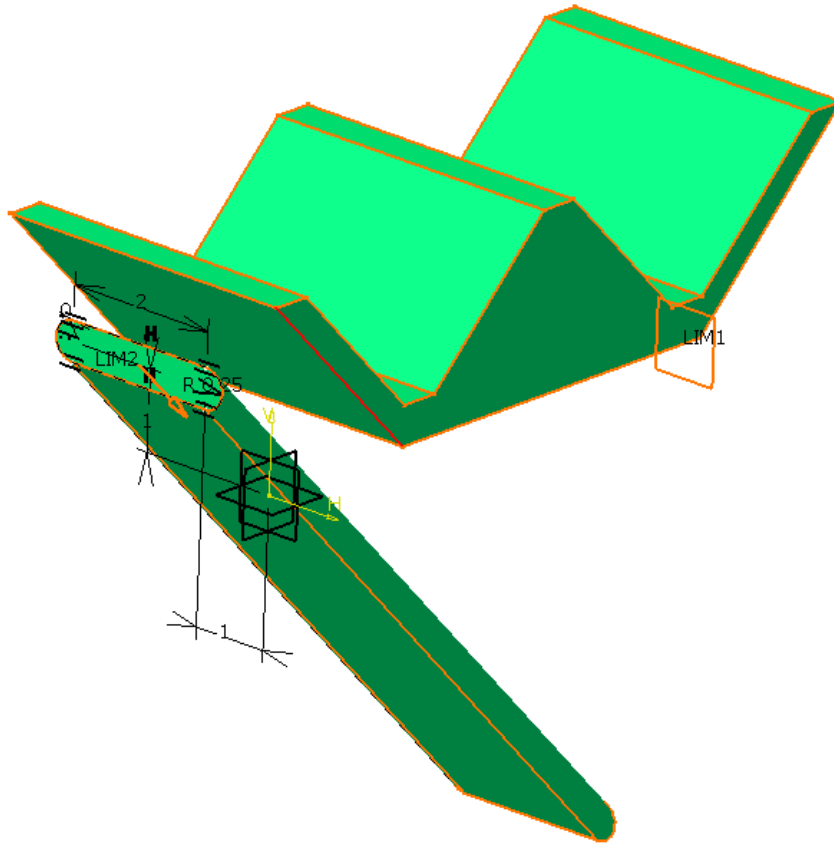


Currently the *Direction* is specified to be *Normal to profile*. You will turn that off and specify an element to be used as the direction. Once again this is just to show you the capabilities of the option.

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

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

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



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

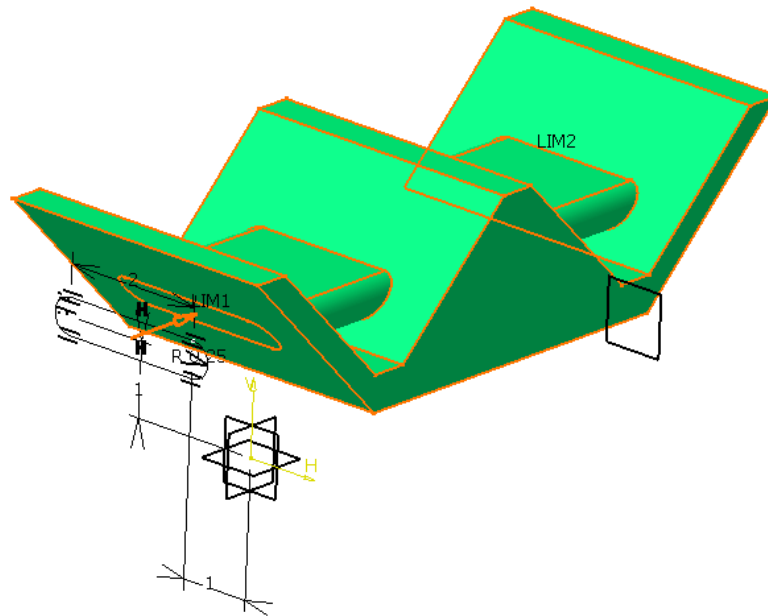
Under the *First Limit* select the *Limit* box. This will allow you to specify a new plane for your limit.

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

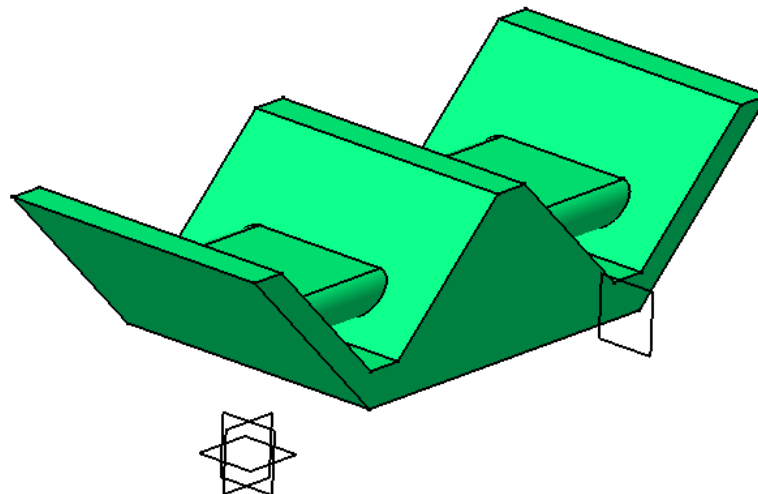
Under the *Second Limit* change the *Type* to *Up to plane*.

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

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



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




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

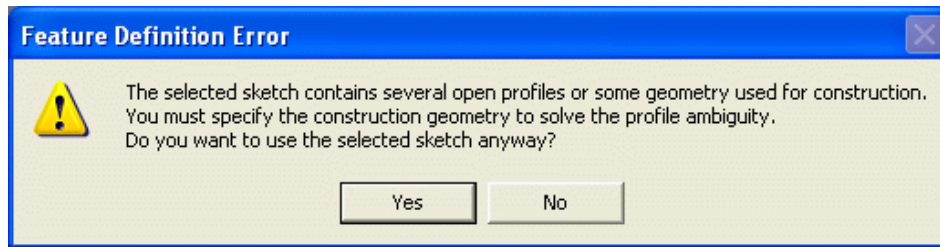
Note: Open profiles (sketches) can be used to create pads or pockets as long as they will be closed by the other faces of your existing part.

Save and close your document.

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

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

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



Select **Yes**.

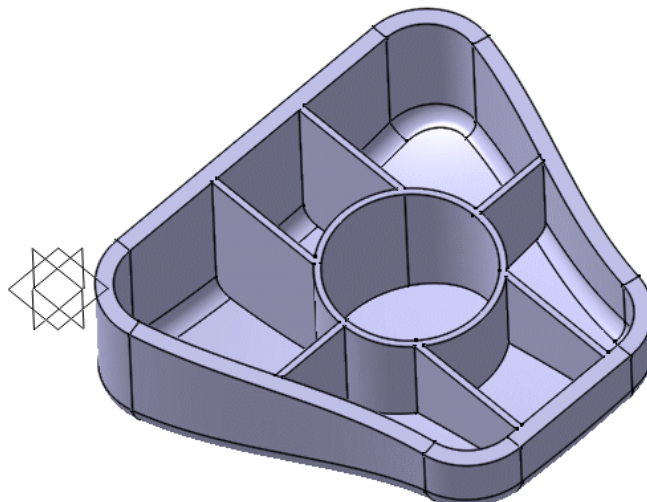
Turn the **Thick** option on. The *Thin Pad* options become available.

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

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


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

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



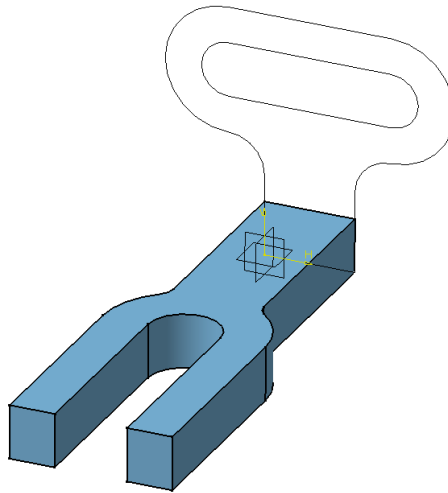
Save and close your document.


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

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

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

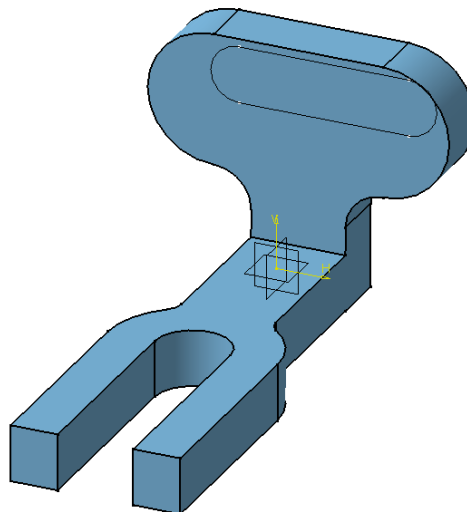
Using the *Type Dimension* and a *Length* of 0.75 create the pad by selecting *OK*. The pad should appear similar to the diagram shown below.



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

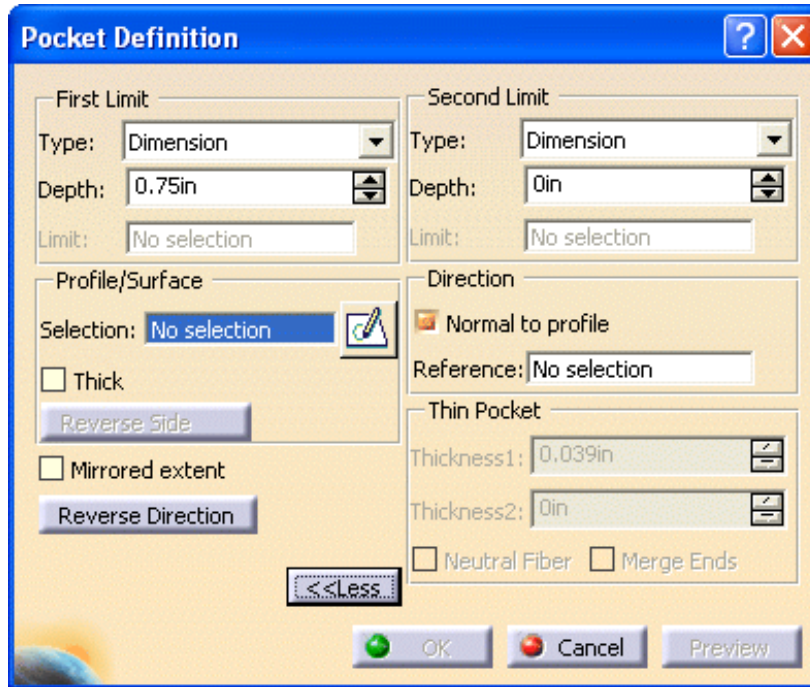
Select *Sketch.2*. This specifies that you want to use that sketch to define the profile of your pad.

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



Pocket

The pocket icon allows you to use a sketch and extrude it in a linear direction producing a pocket. You can create a sketch or profile on-the-fly by pressing the third mouse button while in the *Selection* box. When you create a pocket, a *Pocket Definition* window appears like the one shown below.



Initially the window will appear with only the *First Limit* and then you have the option to select the *More>>* option to see the *Second Limit*. Notice that the options are exactly the same as the options for creating a pad. The major difference between pad and pocket is that a pocket is removed instead of added to your part.

Type


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

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

Profile/Surface

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

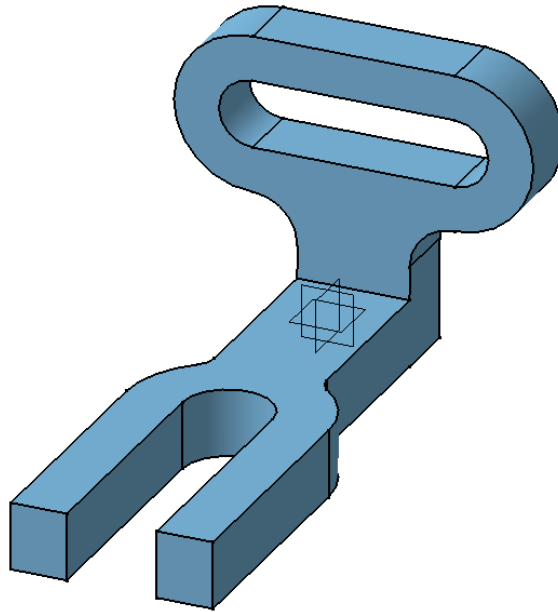
You will now create a pocket in the existing part using *Sketch.3*.

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

Select *Sketch.3*. This specifies that you want to use that sketch to define the profile of your pocket.


Select *Reverse Direction*.

Using the *Type Up to next* create the pocket by selecting *OK*. The pocket should appear similar to the diagram shown below.



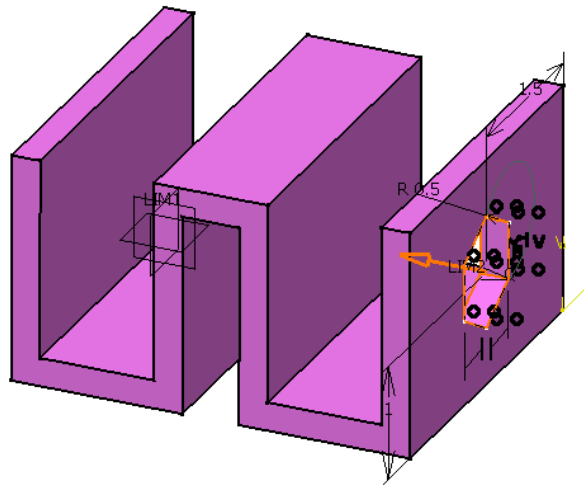
Save and close your document.

Open the **Pocket** document and save with your initials. You should see a pad and a sketch that have been created for you.

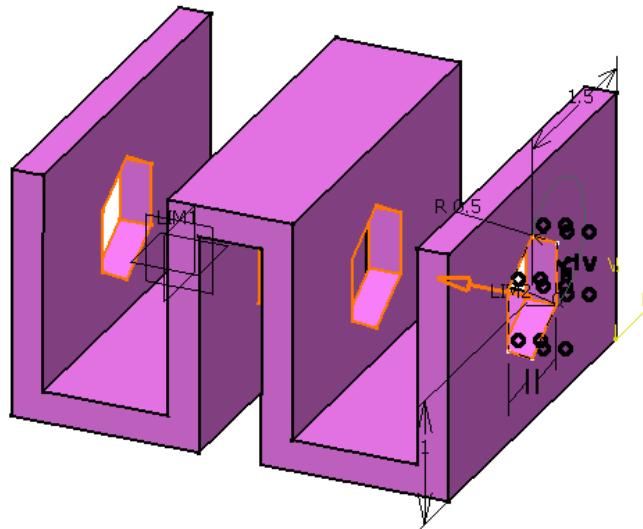
Select the pocket icon.  This will allow you to create a pocket using the sketch.

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

Change the **Type** to **Up to next** and select **Preview**. A preview of the pocket should appear. It should appear similar to the diagram shown below.



Change the **Type** to **Up to last** and select **Preview**. A preview of the pocket should appear. It should appear similar to the diagram shown below.



Select **OK**. The pocket appears.

Save and close your document.

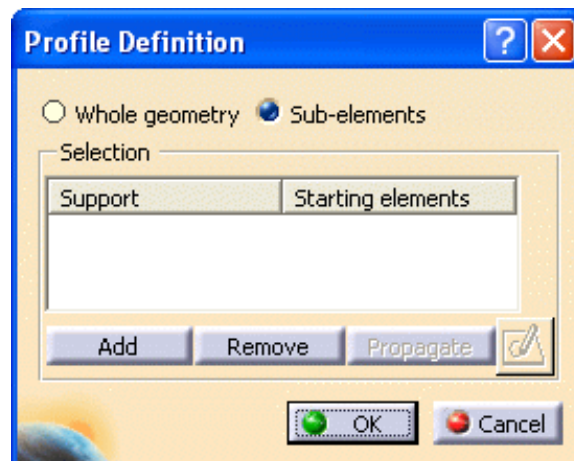
Multiple Profiles

You can create objects using a single profile of a sketch that contains multiple profiles. This allows you to create multiple profiles on the same sketch and then using pad or pocket you can have each profile extrude a different distance as if they were separate sketches. When you do this each pad or pocket will reference the same sketch, just a different part of the sketch.

Open the Multiple profiles document and save with your initials. You should see a sketch that has already been created for you.

Select the pad icon. 

In the Selection box press the third mouse button and select *Go to profile definition*. The *Profile Definition* window appears as shown below.



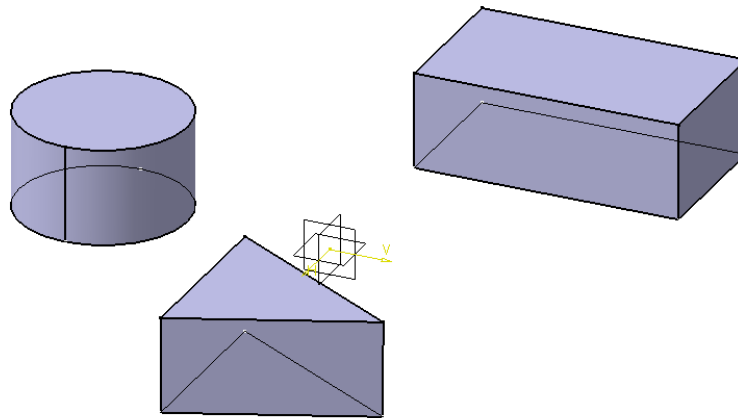
This will allow you to select the sub-elements of a sketch.

Select the box portion of the sketch. Notice just by selecting an edge that entire profile was selected but not the whole sketch.

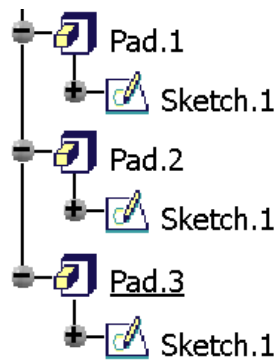
Select OK. This closes the *Profile Definition* window and returns to the *Pad Definition* window with the *Selection* being *Complex*.

Change the Length to be 1.0 and select OK. The pad is created using just the one profile of the sketch. Notice that the sketch still remains shown because you did not use the entire sketch therefore it did not automatically hide.

Repeat the steps and create pads out of the other two profiles 1.0 inch in length. It should appear similar to the diagram shown below.



The specification tree should have each pad using the same sketch as shown below.



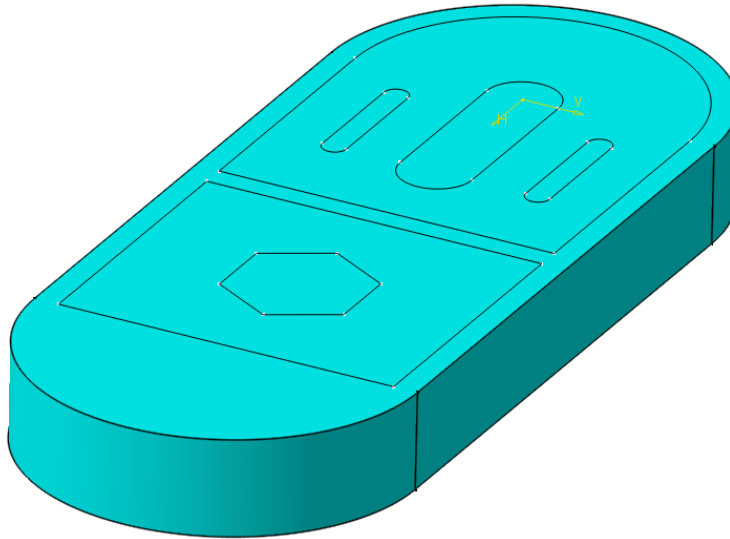
It is normally a better idea to create a separate sketch for each object but there are times when this capability becomes extremely useful, especially when constraining.

Save and close your document.

Multi-Pad and Multi-Pocket

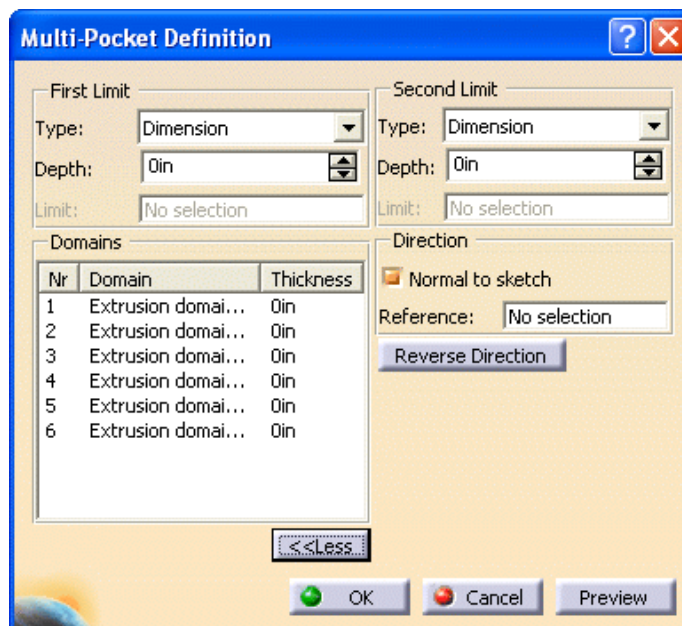
There is another option to creating a pad or pocket with a sketch that contains multiple profiles. Each profile can have its own distance but, instead of using a portion of a sketch for each pocket or pad you can create one pad or pocket using all the different profiles of the sketch. These are referred to a multi-pads or multi-pockets. In this example you will be creating a multi-pocket but both options work very similarly.

Open the Multipocket document and save with your initials. A part should appear with a sketch that contains multiple profiles as shown below.



Select the multi-pocket icon.  This is located under the pocket icon.

Select *Sketch.2*. A *Multi-Pocket Definition* window appears. It is shown below with the *More>>* button selected.



All of the options are the same as they were for a normal pocket except that the *Type* options are limited to *Dimension* only for now. It is expected that additional options will appear here in the future. Notice that all six profiles appear under the *Domains* area. You can select a domain and then specify a *Depth* for that domain. You also have the option of selecting multiple domains and entering the depth only once.

Select the *Extrusion domain.1* from the window. It highlights. You can now enter a depth.

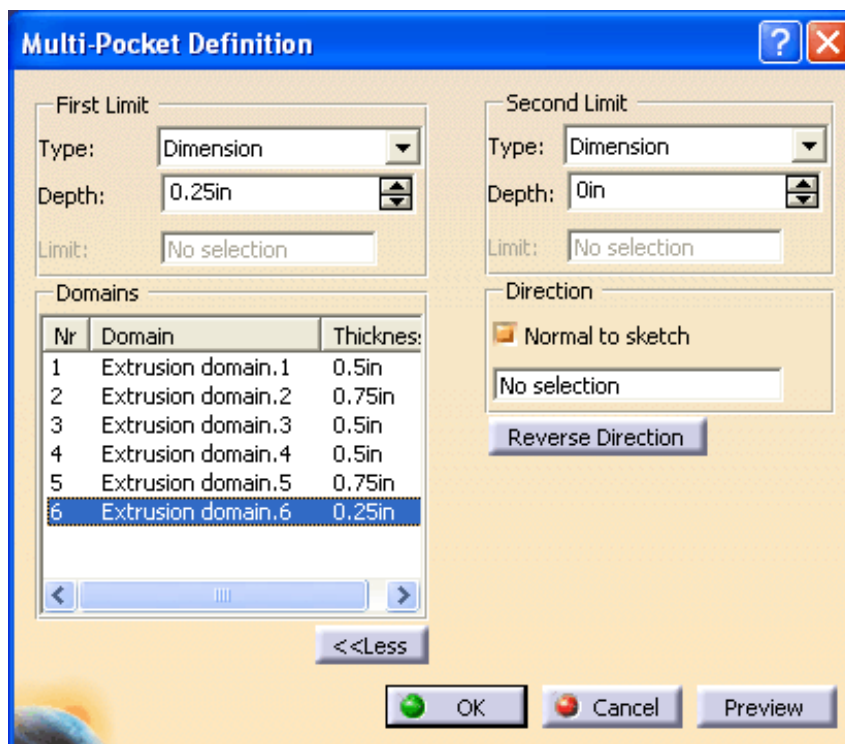
Key in 0.5 for the *Depth*. Do not press the Enter key or it will go ahead and build the pocket.

Select the *Extrusion domain.2* from the window and key in 0.75 for the *Depth*.

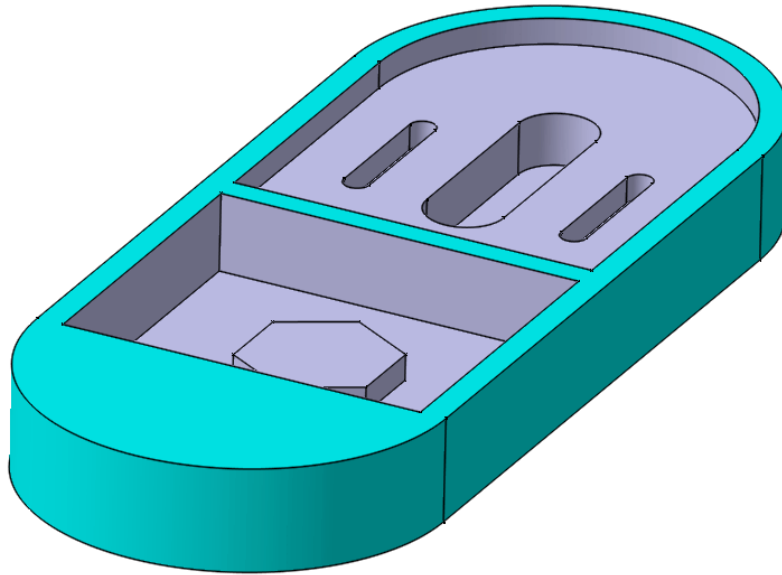
Select the *Extrusion domain.3* and *Extrusion domain.4* from the window. You can do this by holding down the Ctrl key when you select the second domain.

Key in 0.50 for the *Depth*.

Key in 0.75 for the *Depth* of *Extrusion.5* and 0.25 for the *Depth* of *Extrusion.6*. The window should appear similar to the one shown below.



Select OK. The multi-pocket is created. It should appear similar to the one shown below.



Notice that this is different from using multiple profiles to create multiple pads or pockets. This method creates just one multi-pad or multi-pocket using the one sketch. If you make changes to the sketch that was used for the multi-pocket then you will have to modify the multi-pocket.

Save and close your document.

