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Introduction

CATIA Version 5 Sheet Metal Design

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

- Defining sheet metal parameters
- Creating sheet metal parts
- Performing operations on sheet metal parts
- Modifying sheet metal parts
- Working on sheet metal parts in both the folded and unfolded views
- Recognizing normal parts as sheet metal parts

Sheet Metal

Most parts can be created using just the Part Design and Wireframe and Surfaces tools; however, there are times when sheet metal specific parts are needed. The sheet metal workbenches allow you to create parts that can be folded and unfolded in order to make them easier to work with. You will find that many times using the sheet metal features can simplify your design. There are a few workbenches in CATIA V5 that deal with sheet metal options. Many of these options appear in more than one of the sheet metal workbenches. They will only be covered once. This course will cover all of the options found in the Generative Sheet Metal Design and Aerospace Sheet Metal Design workbenches.

Sheet Metal Terminology

This section will discuss some standard sheet metal terminology and definitions.

Bend Tangent Line (BTL) Represents the locations where the bend begins and ends

Formed Block Line (FBL) Represents the intersection of the two inner faces on either side of the bend

Inside Mold Line (IML) Represents the intersection of the two inner faces on either side of the bend. For a formed part, the IML and the FBL are the same.

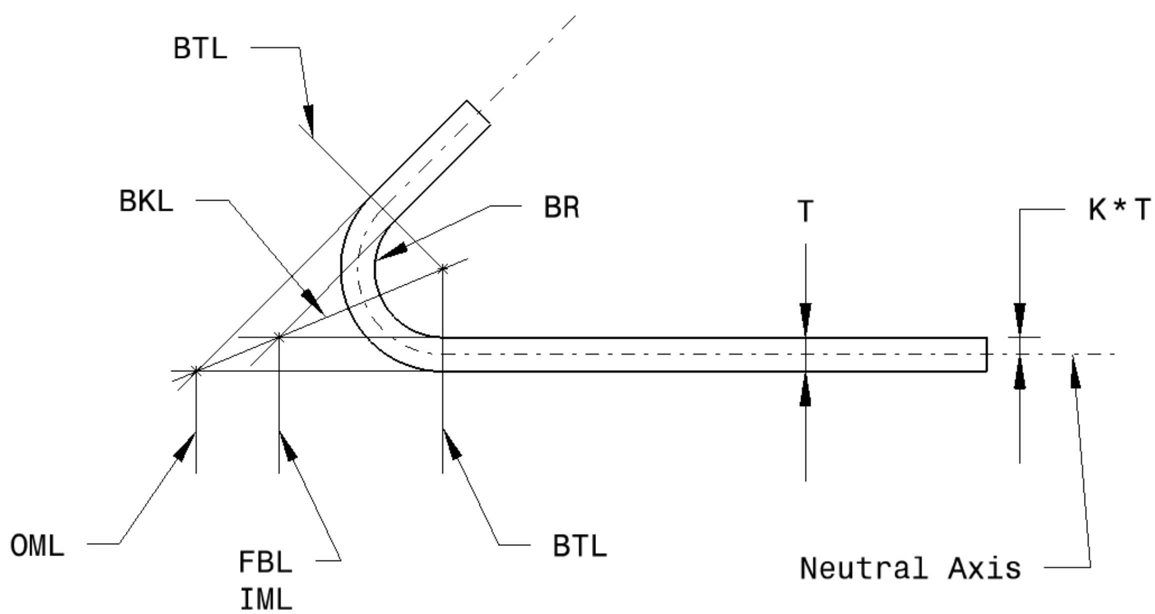
Outside Mold Line (OML) Represents the intersection of the two outer faces on either side of the bend

Break Line (BKL) Represents the intersection of the center of the bend and the neutral axis

Thickness (T) Defines the thickness of the part

Bend Radius (BR) Defines the radius of the inside of the bend

K Factor (K) Represents the location of the neutral axis with respect to the thickness and bend radius of the part. As defined by the DIN 6935 Standard, K = (0.65+Log(R/T)/2)/2. The K factor can be approximated by 0.447T if necessary.

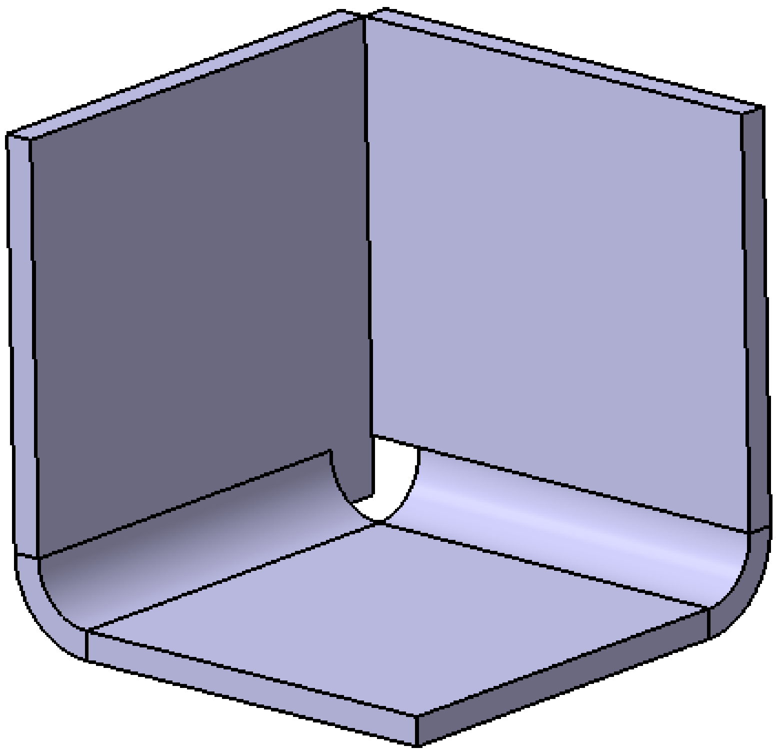


Corner Relief

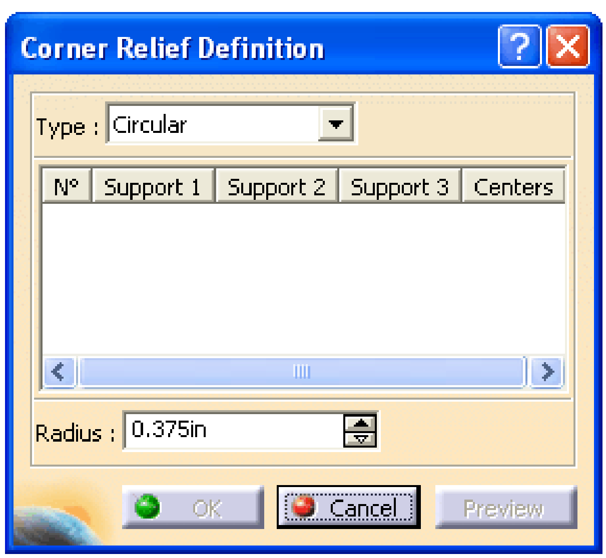
This option will allow you to create corner reliefs where two bends come together.

Open the Corner Relief document. This part is made up of three walls with two bends between the bottom wall and the two side walls.

Notice that the corner is very ugly. Since no corner relief has been defined, CATIA had to guess at how the corner should be handled when creating the two bends.

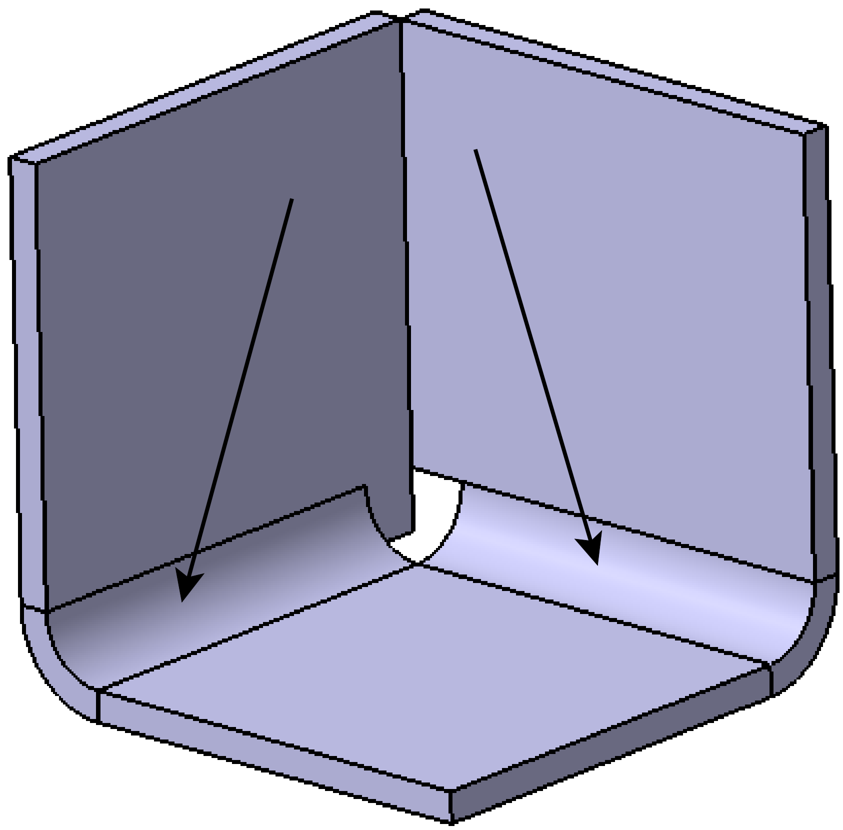


Select the Corner Relief icon.  The *Corner Relief Definition* window appears.

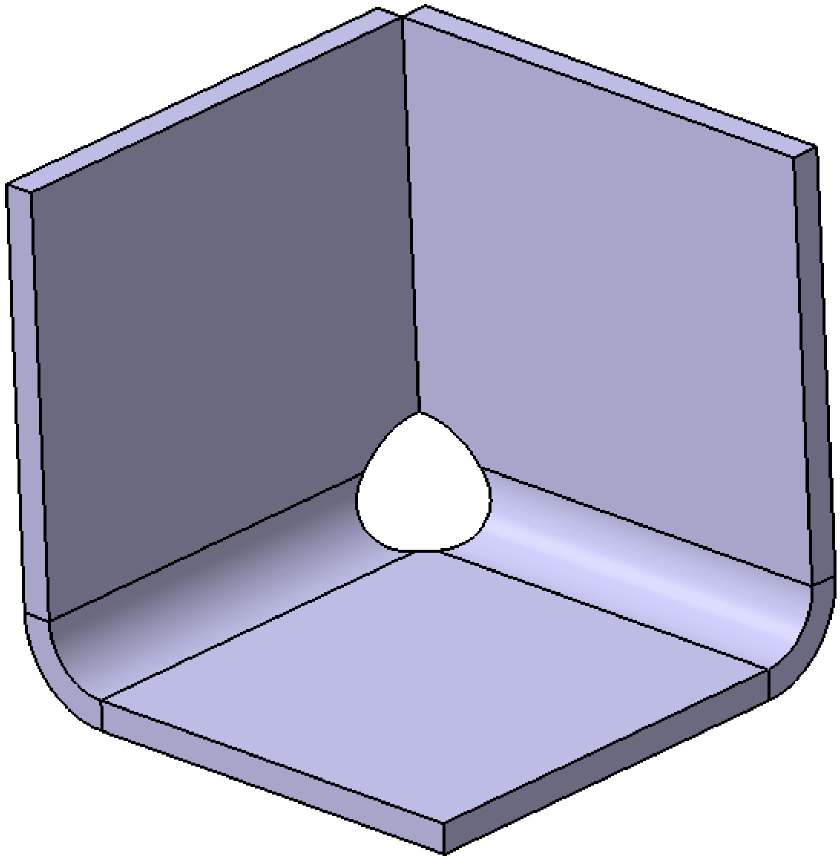


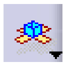
Type	Specifies whether corner relief will be *Circular*, *Square* or a *User Profile*.
Support 1, 2, 3	Specifies the supports for the corner relief
Centers	Defines the center location for a *Circular* profile
Radius	Defines the radius for a *Circular* profile

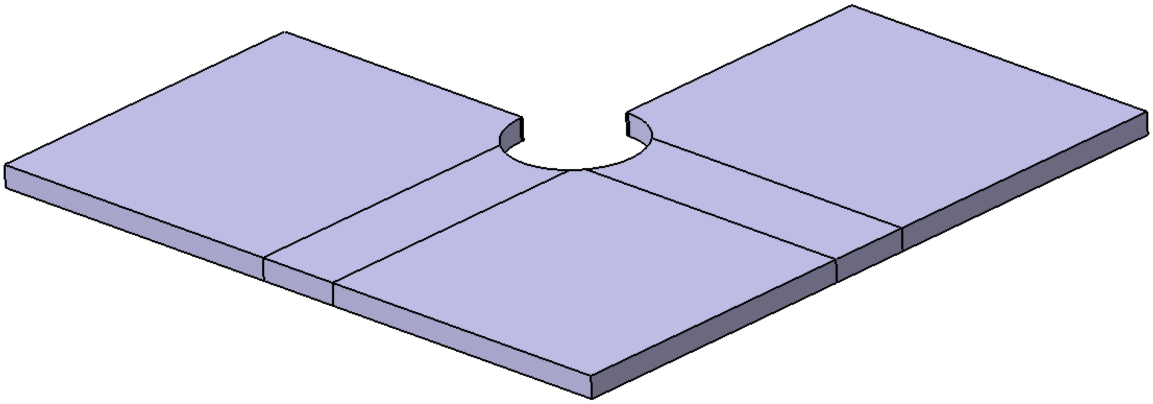
Select the two bend faces as shown below to define two supports for the relief.



Set the *Radius* to be 0.375 and select *OK*. The corner relief is created.

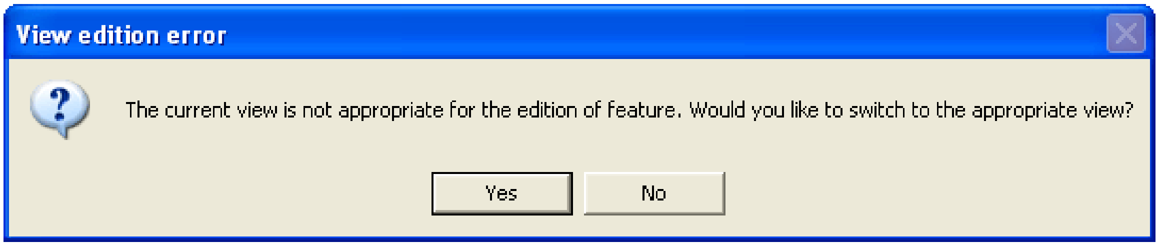


Select the Fold/Unfold icon.  The part is unfolded.



Show *Sketch.4* from the specification tree. You will replace the circular corner relief with a square profile corner relief using *Sketch.4*.

Double select *Corner Relief.* in the specification tree. A *View edition error* window appears.



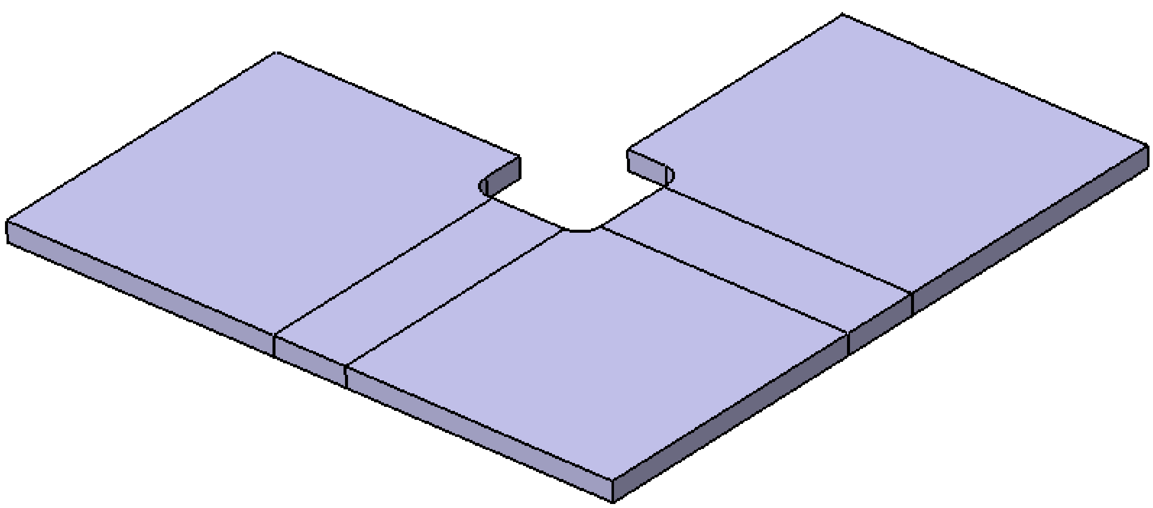
Select *Yes*. The *Corner Relief Definition* window reappears.

Change the *Type* to be *User Profile* from the window. A *User profile error* window appears.

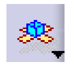
Select *Yes*. The part is unfolded. It depends on the profile type that you are using as to whether the part needs to be folded or unfolded.

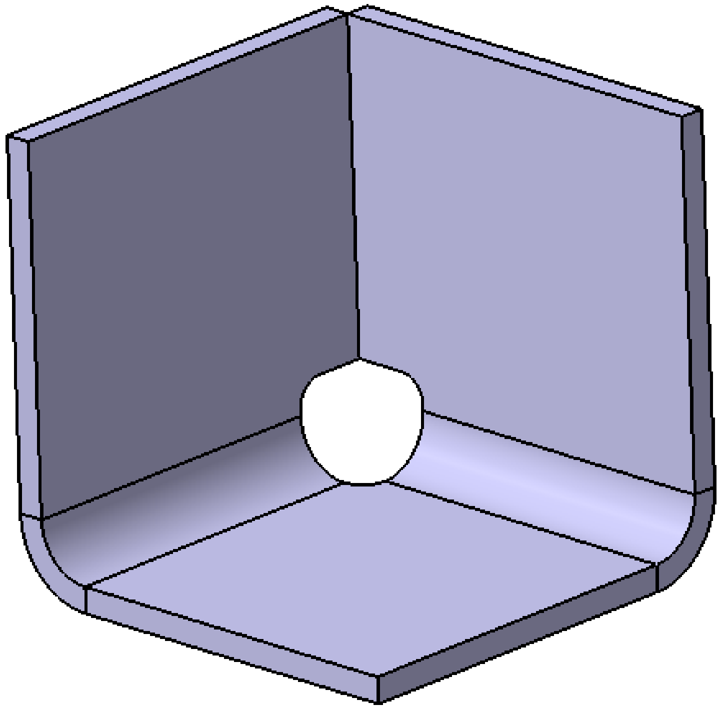
Select in the *Profile* selection box in the window and select *Sketch.4* from the specification tree to define the profile for the corner relief operation to use.

Select the two bend faces again to define the supports and select *OK*. The corner relief is created as shown. Now it uses the profile defined in *Sketch.4* instead of a circular profile.



Hide *Sketch.4*.

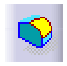
Select the Fold/Unfold icon.  The part is folded again.

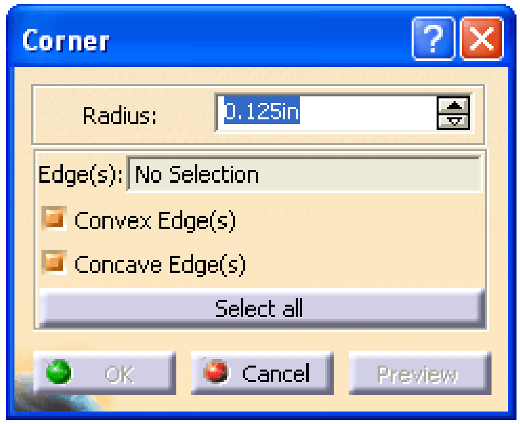


You will use this model for the next exercise.

Corner

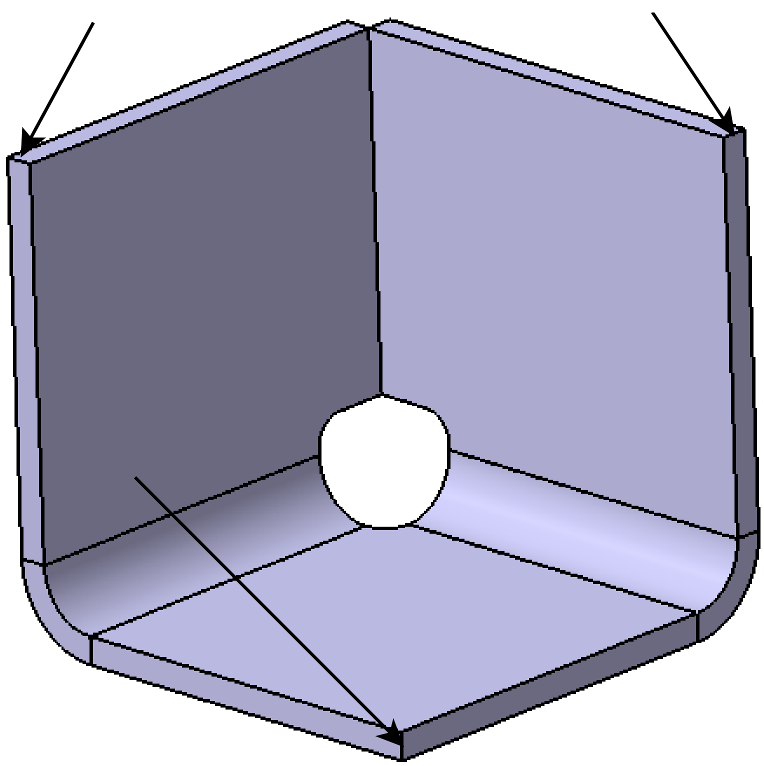
The corner function is much like the fillet function in the Part Design workbench. This option allows you to round off corners on sheet metal parts.

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

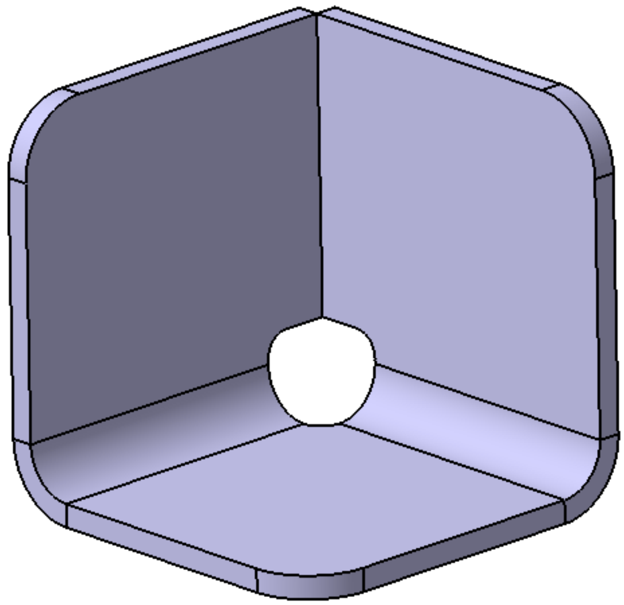


Radius	Specifies the size of the corner
Edges	Specifies the edges that are to be rounded off
Select all	Allows you to select all available edges for the corner operation

Select the edges as shown below to define the *Edges* for the corner operation.



Set the *Radius* to be 0.375 and select *OK*. The model appears as shown.



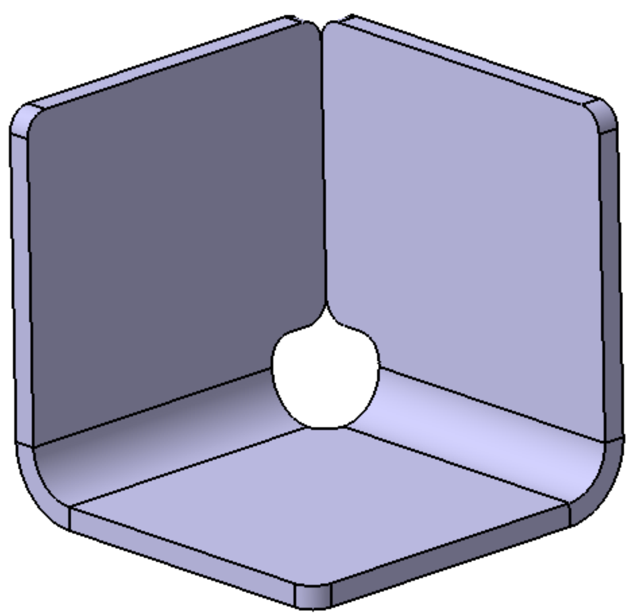
Double select on *Corner.1* in ths specification tree. The Corner Definition window reappears.

Change the *Radius* to 0.125.

Select the *Cancel selection* button in the window. The three edges that were selected are now deselected.

Select the *Select all* button in the window. Notice CATIA selects seven edges that are available to be rounded off.

Select *OK*. The model appears as shown.

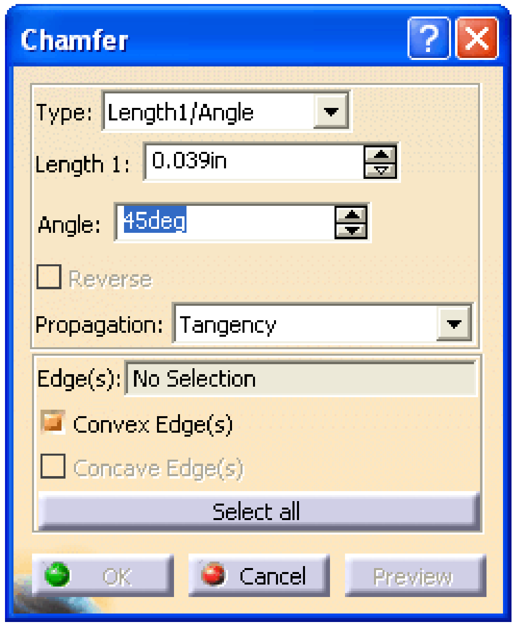


Delete *Corner.1* from the specification tree. You will use the same model for chamfers.

Chamfer

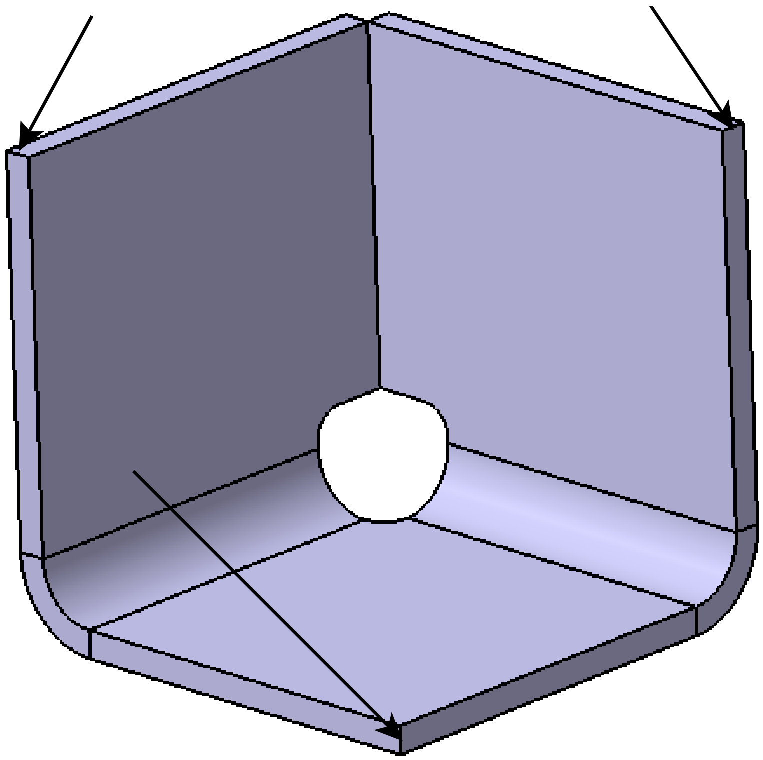
This option is much like the chamfer function in the Part Design workbench. Chamfer allows you to chamfer off corners on sheet metal parts.

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

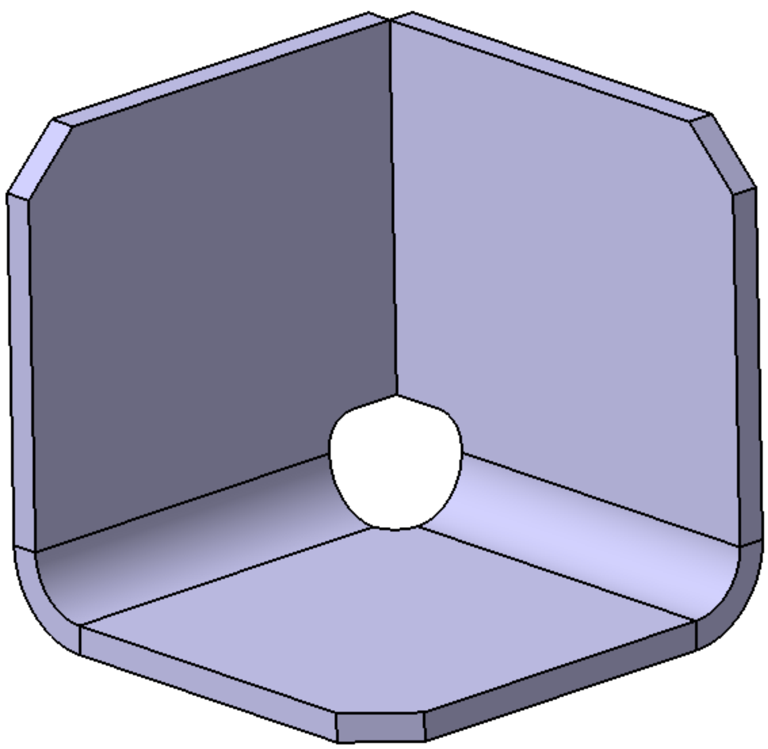


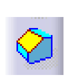
Type	Specifies how the chamfer will be defined
Length	Specifies the length of the chamfer
Angle	Specifies the angle of the chamfer
Reverse	Allows you to reverse the orientation of the chamfer
Propagation	Specifies whether the chamfer will propagate around tangent edges or not
Edges	Specifies the edges that are to be chamfered
Select all	Allows you to select all available edges for the chamfer operation

Select the three edges as shown below.

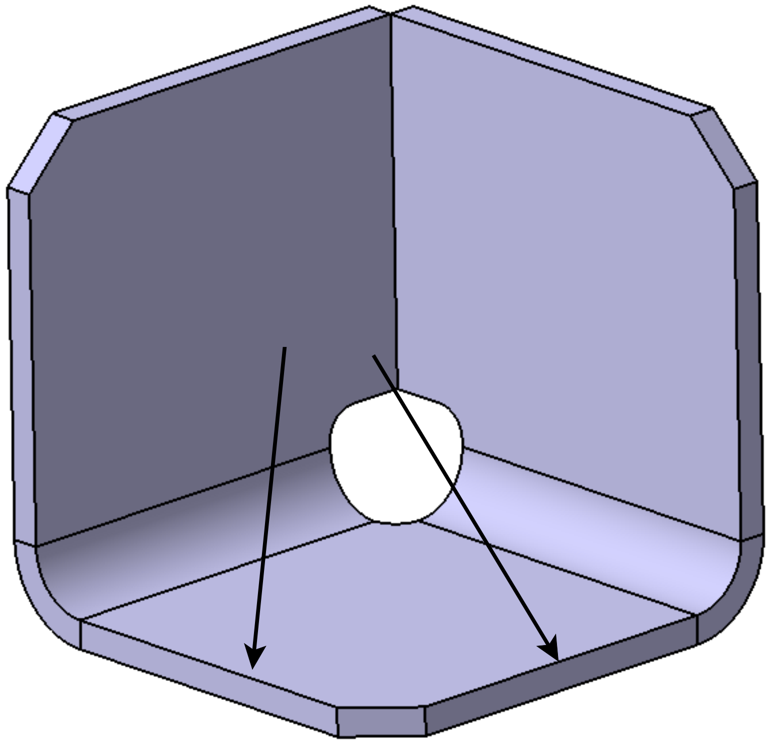


Set the *Type* to be *Length1/Angle* and set *Length 1* to be 0.25 and the *Angle* to be 45 and select *OK.* The chamfer is created.

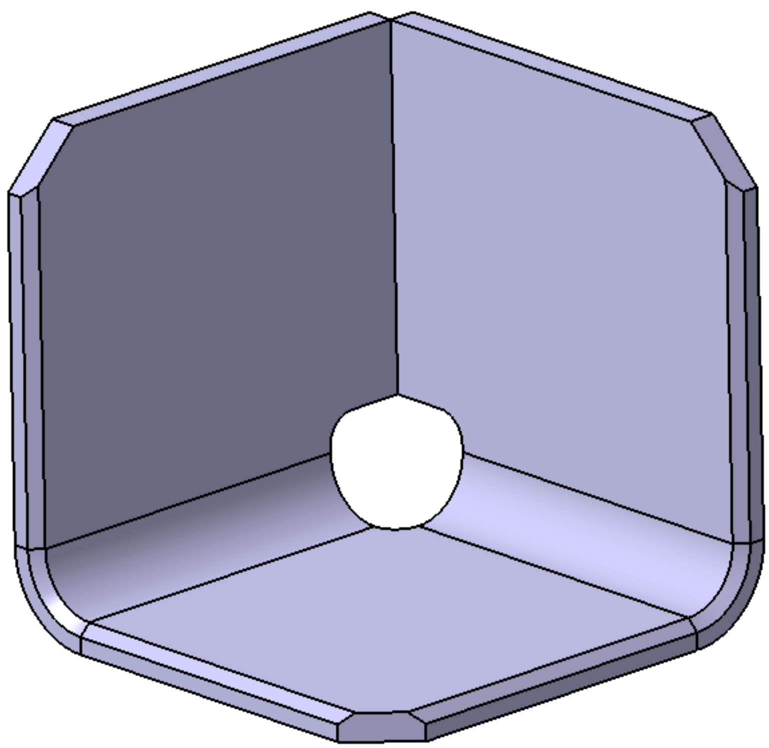


Select the Chamfer icon again.  The *Chamfer Definition* window appears.

Change *Length 1* to be 0.05 and select the two edges as shown below. Make sure *Propagation* is set to *Tangency*.



Select *OK*. The chamfer is created.



Notice the chamfer propagates around the tangent edges.

Save and close the document.

Stamps

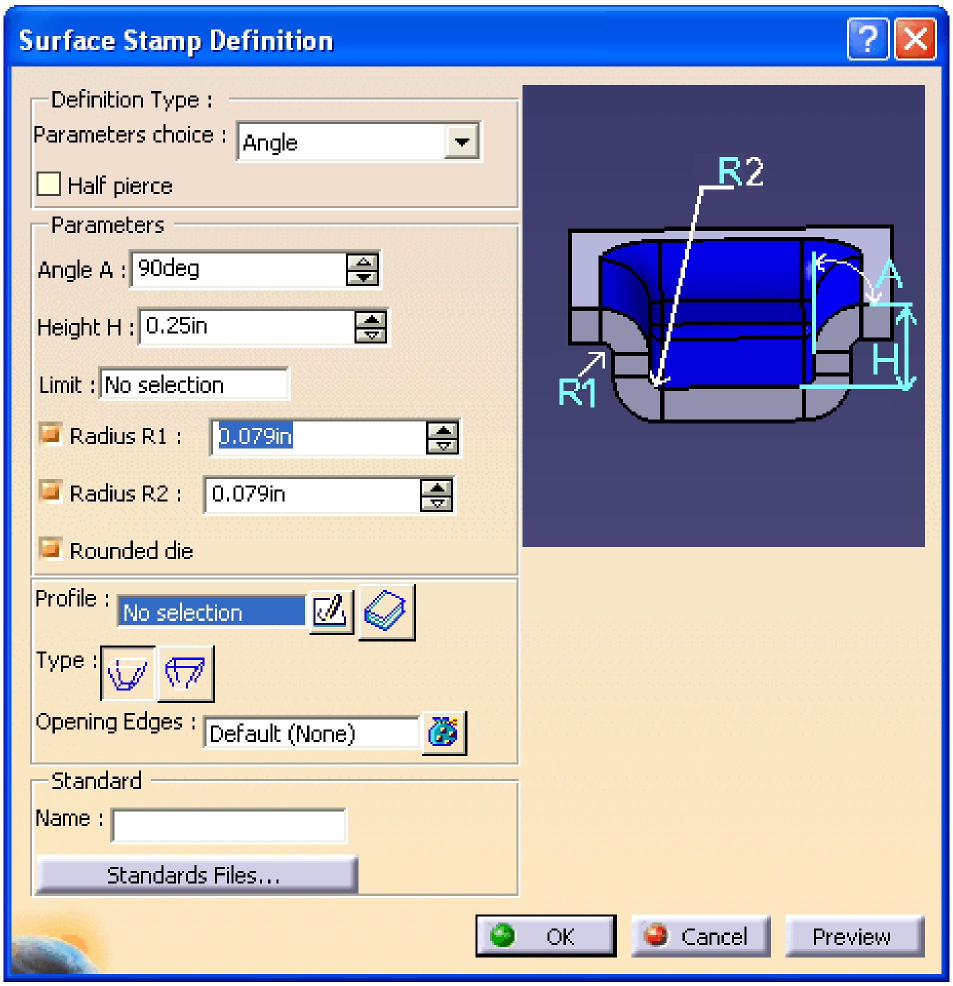
There are number of different options available to create stamps within a sheet metal part.

Surface Stamp

This option allows you to stamp a surface using a profile to define the shape of the stamp.

Open the Surface Stamp document. There are a number of sketches already created.

Select the Surface Stamp icon.  The *Surface Stamp Definition* window appears.



Definition Type

Parameters choice	Specifies the parameters that will be used to define the stamp
Half pierce	Specifies that the stamp will half pierce the part

Parameters

Angle	Defines the angle of the stamp
Height	Defines the height of the stamp

Limit	Specifies a limit for the stamp
Radius R1	Specifies the radius around the top of the stamp
Radius R2	Specifies the radius around the bottom of the stamp
Rounded die	Specifies the radius around the vertical edges of the stamp

Profile Specifies the shape of the stamp

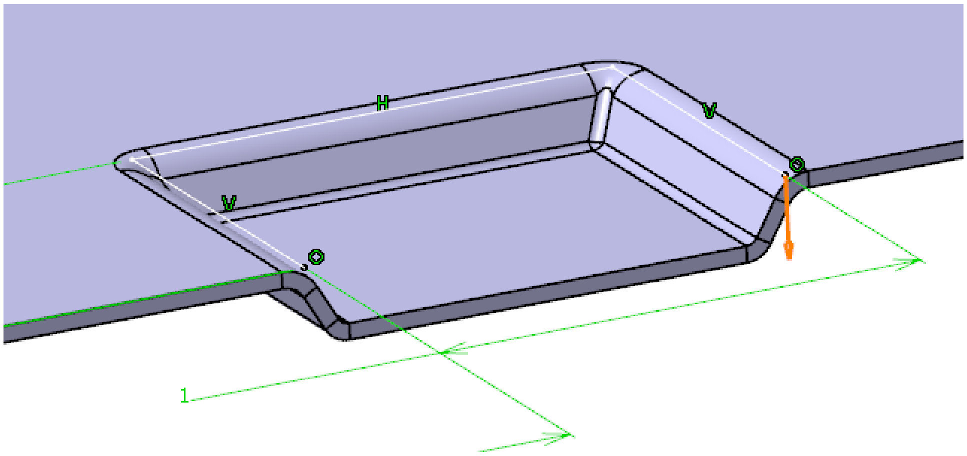
Type Determines whether the profile specifies the size of the stamp at the top or the bottom

Opening Edges Specifies any open edges in the profile

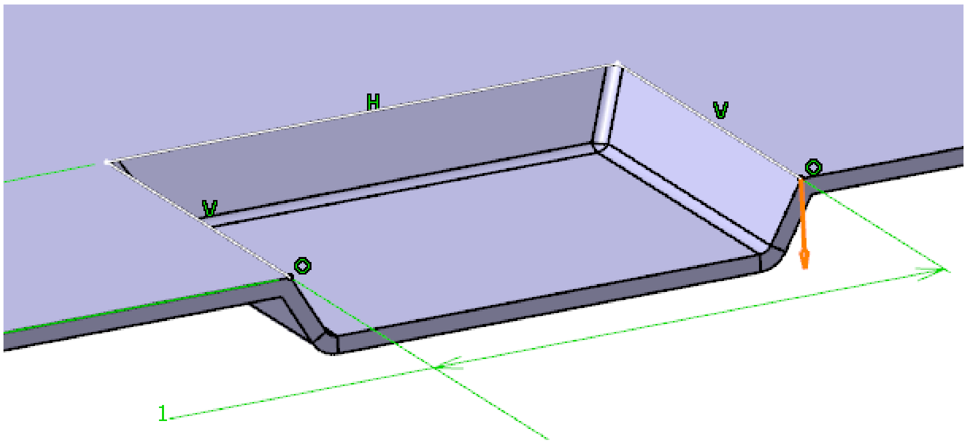
Standard Allows you to use a specify a file to define the standards

Select *Sketch.2* to specify the *Profile* for the stamp.

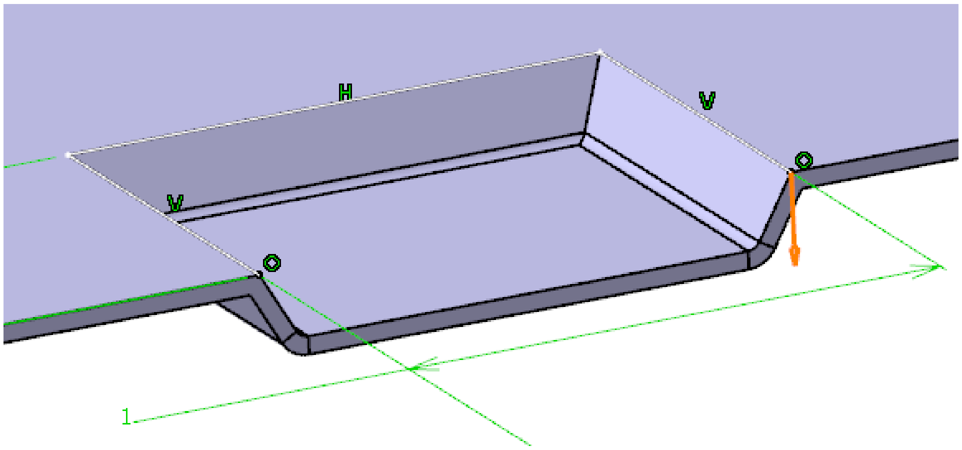
Set the *Angle* to be 60, the *Height* to be 0.125, *Radius R1* to be 0.05 and *Radius R2* to be 0.025 and select *Preview*. The stamp should appear as shown.



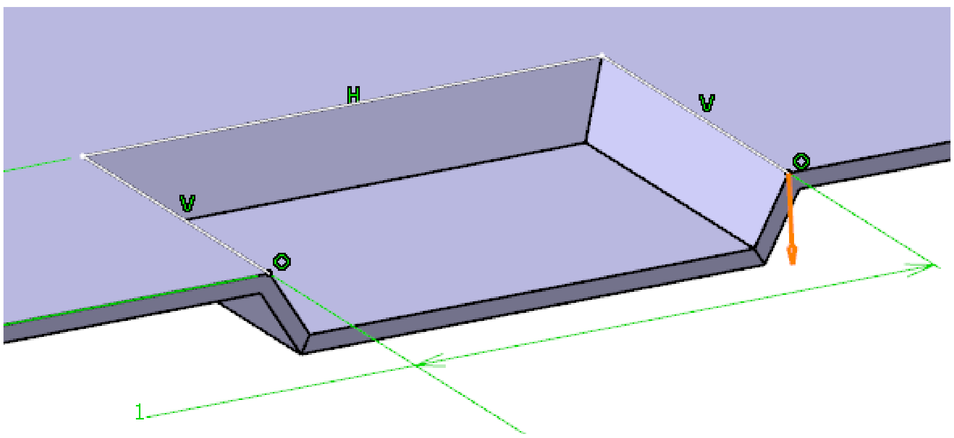
Turn off *Radius R1* and select *Preview*. The stamp appears as shown. Notice the upper radius was turned off.



Turn off *Rounded die* and select *Preview*. The stamp appears as shown. The vertical corners are eliminated.

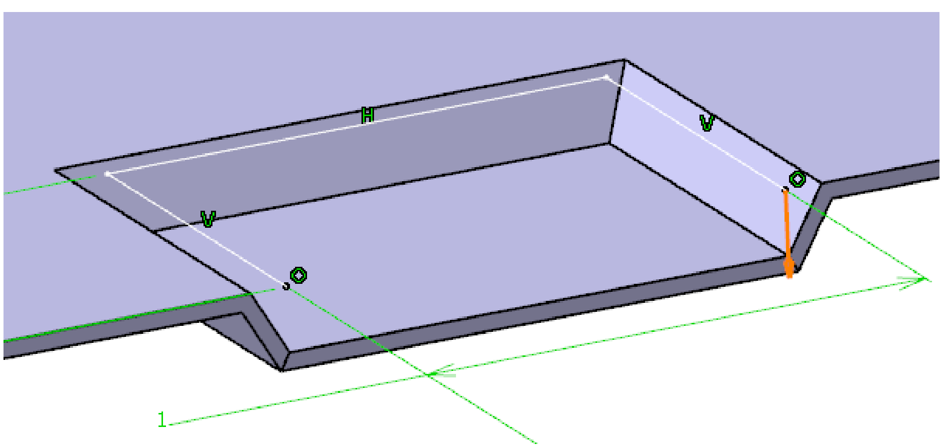


Turn off *Radius R2* and select *Preview*. The stamp appears as shown. The lower radius is deactivated.



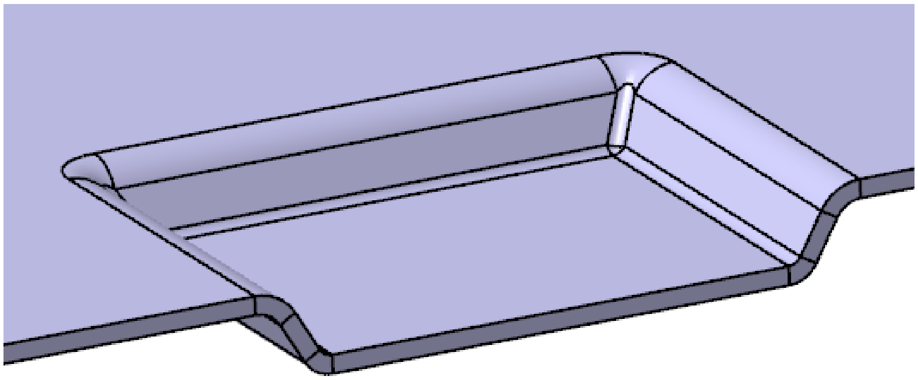
You should also notice that the upper edge of the stamp is located directly in line with the sketch profile. This is because the *Type* is set to Upward Sketch Profile by default. 

Select the Downward Sketch Profile to define the *Type* and select *Preview*.  The stamp is modified so that the lower edge is aligned with the sketch profile.



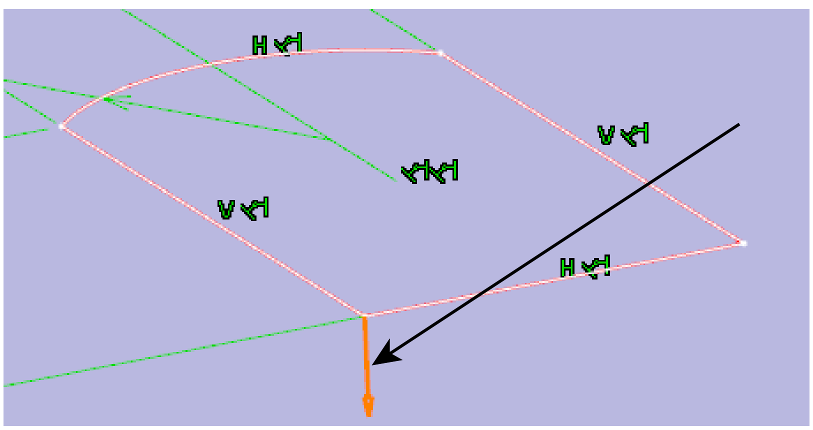
Select the Upward Sketch Profile icon to switch the *Type* back. 

Turn on *Radius R1*, *Radius R2* and *Rounded die* and select *OK*. The stamp appears as shown.

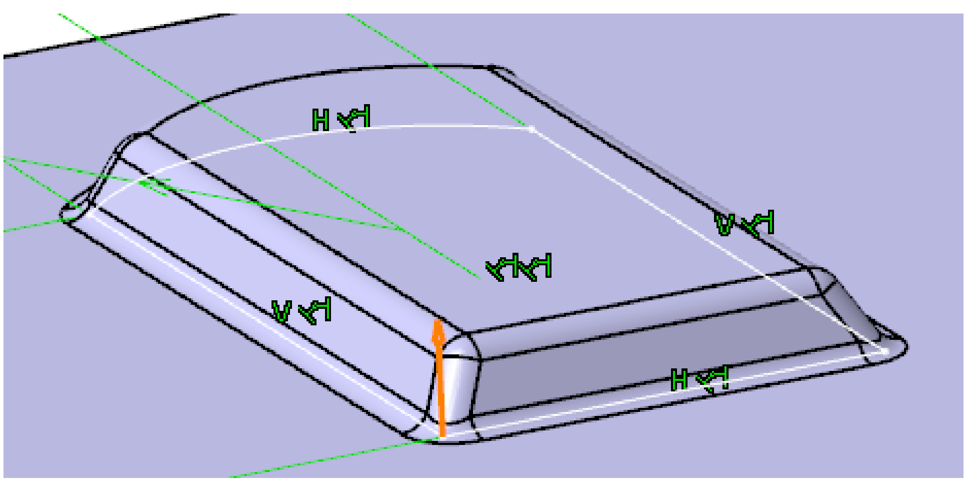


Select the Surface Stamp icon again.  This time you will use a closed profile.

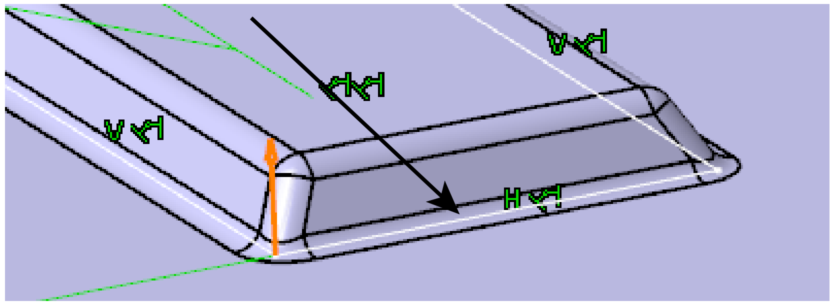
Select *Sketch.3* to define the *Profile* for the stamp and select the arrow in the display as shown to switch the direction of the stamp.



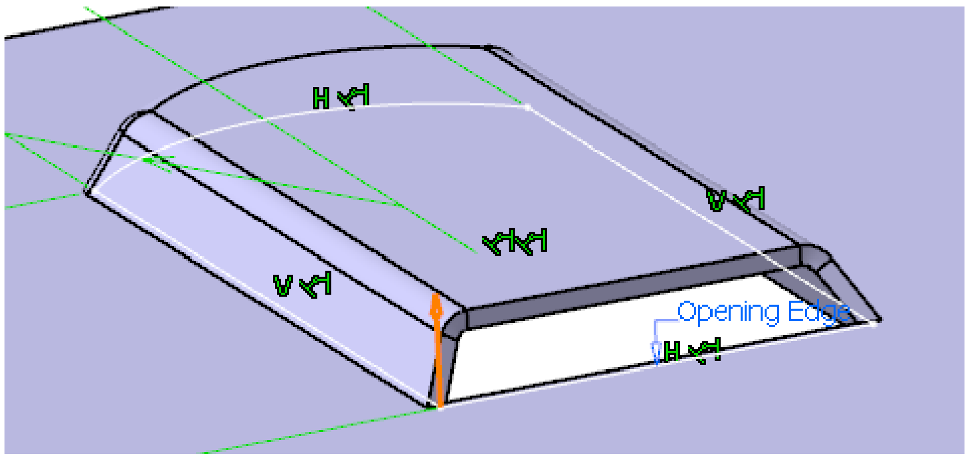
Select *Preview*. The stamp appears as shown.



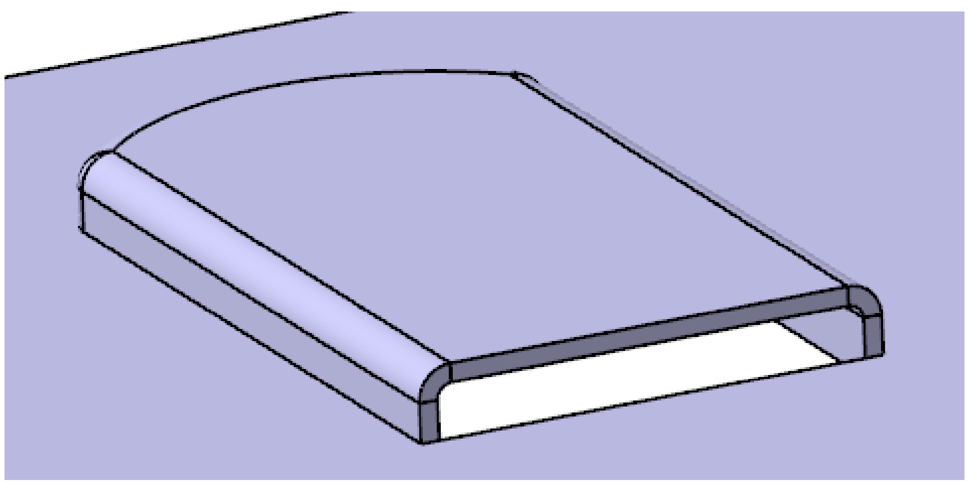
Select in the *Opening Edges* selection box and select the edge of the sketch as shown below. You are selecting the line from the sketch.



Turn off *Radius R1* and select *Preview*. The stamp appears as shown.

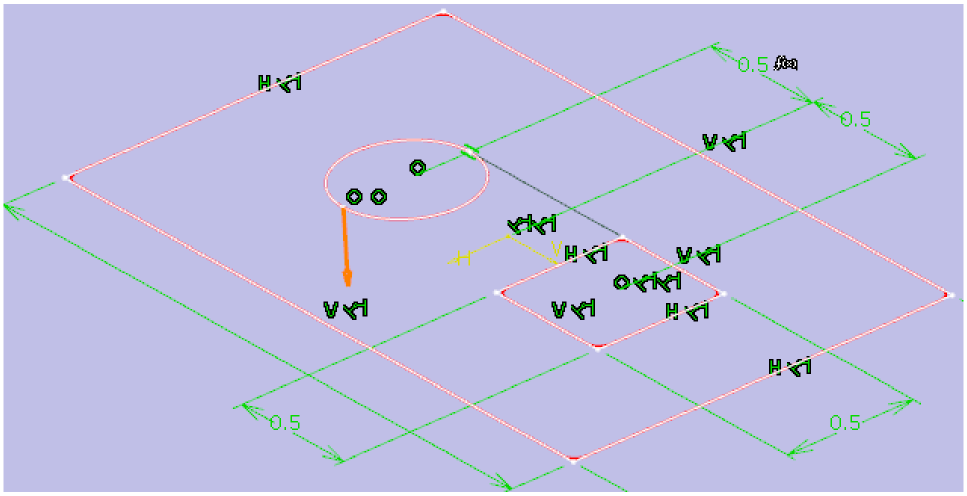


Change the *Angle* to 90 and select *OK*. The stamp is created.



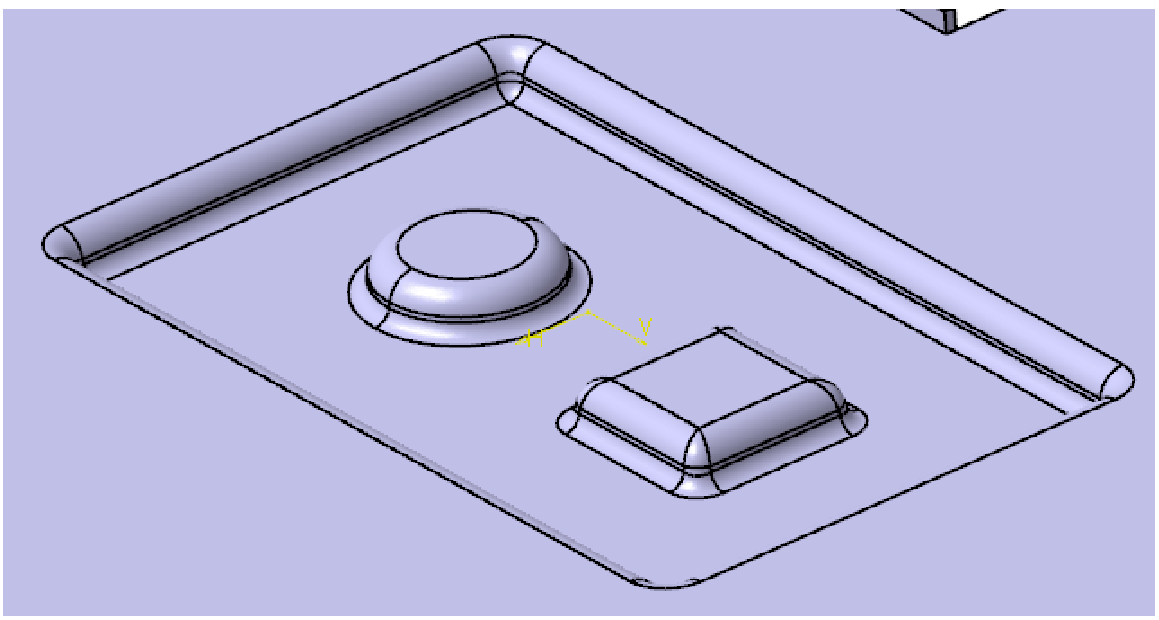
Select the Surface Stamp icon again.  This time you will use a sketch with multiple profiles.

Select *Sketch.8* to define the *Profile* for the stamp. Notice the sketch contains three closed profiles.



Set the *Angle* to be 80 and both *Radius* parameters to be 0.05.

Select *OK*. The stamp appears as shown.

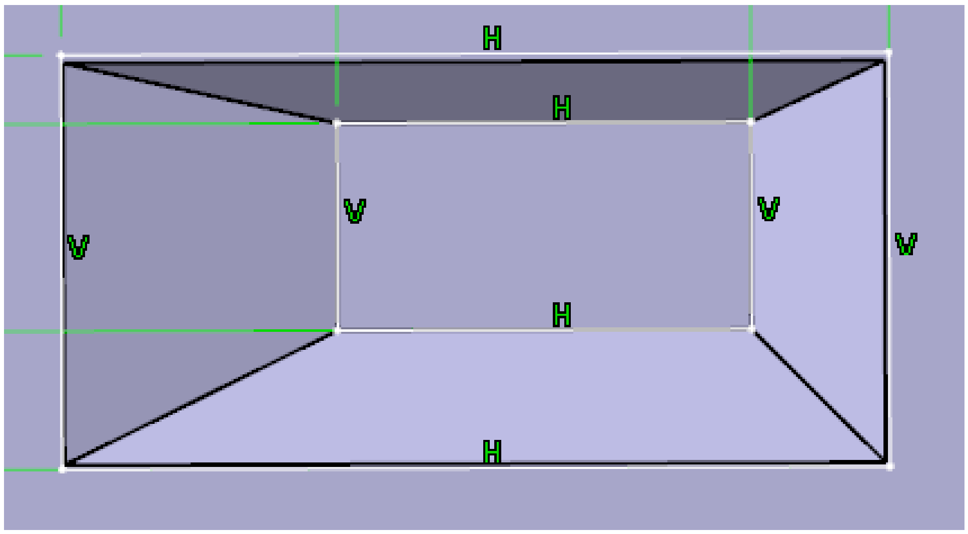


Using multiple profile sketches allows you to create more complicated surface stamps.

Select the Surface Stamp icon again.  This time you will use the *Punch & Die* option.

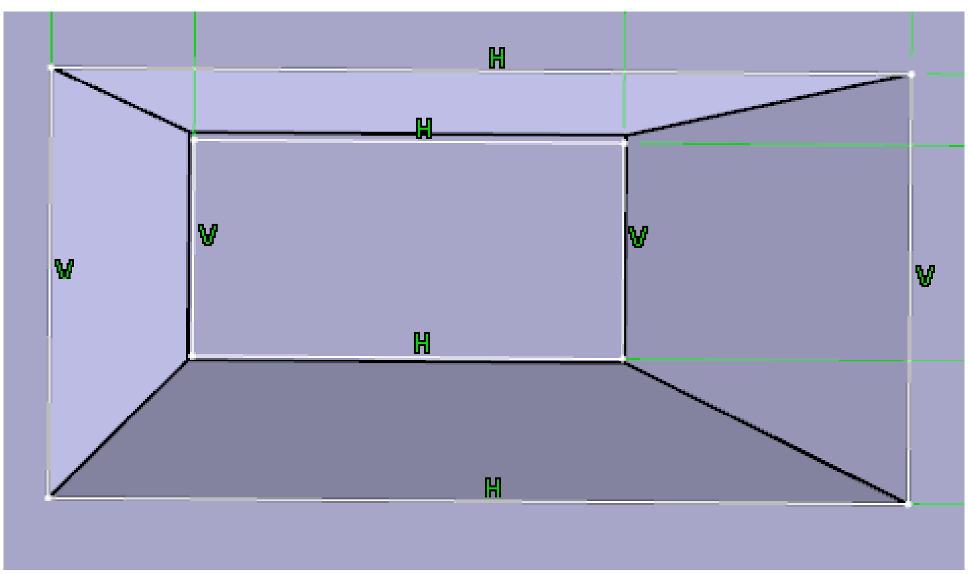
Select *Sketch.5* to define the *Profile* for the stamp and switch the *Parameters choice* to be *Punch & Die*. Notice the sketch contains two closed profiles. The exterior profile will behave as the die and the interior profile will act as the punch.

Set the *Height* to 0.125, turn off *Radius R1* and *Radius R2* and select *Preview*. If you looks straight down on the top of the stamp it should appear as shown.



Notice the inner profile of the sketch matches up with the bottom edges of the stamp. This is because the inner profile of the sketch is acting as a punch for the stamp.

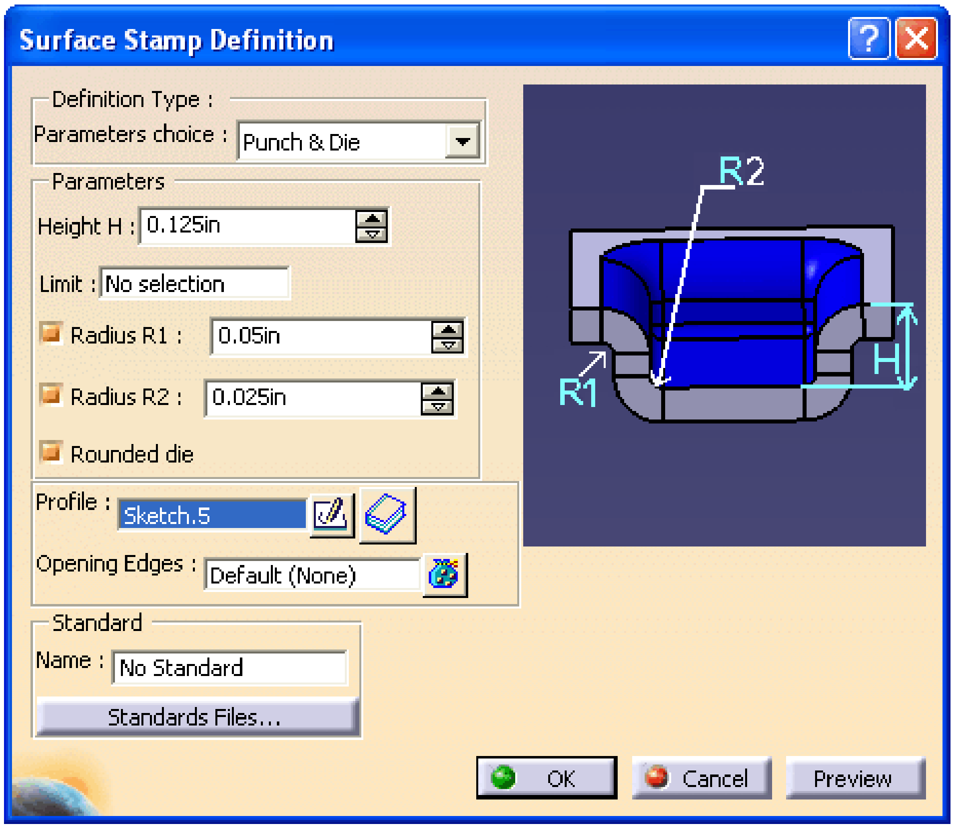
If you looks straight down on the bottom of the stamp it should appear as shown.



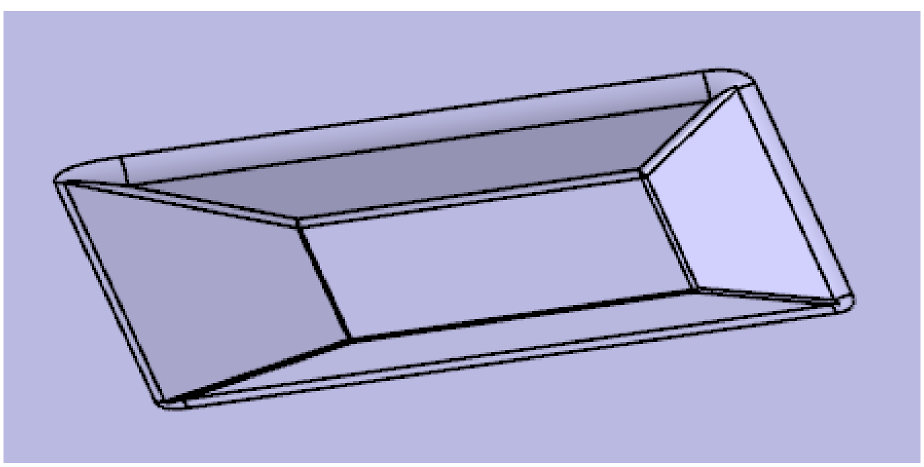
Notice the outer profile of the sketch matches up with the bottom outer edge of the stamp. This is because the outer profile of the sketch is acting as a die for the stamp.

You should also see that the angles of the stamp are controlled by the two profiles of the sketch. The profiles must be similar shapes, but they do not have to be centered on one another as you can see in this example.

Set the options in the window as shown.



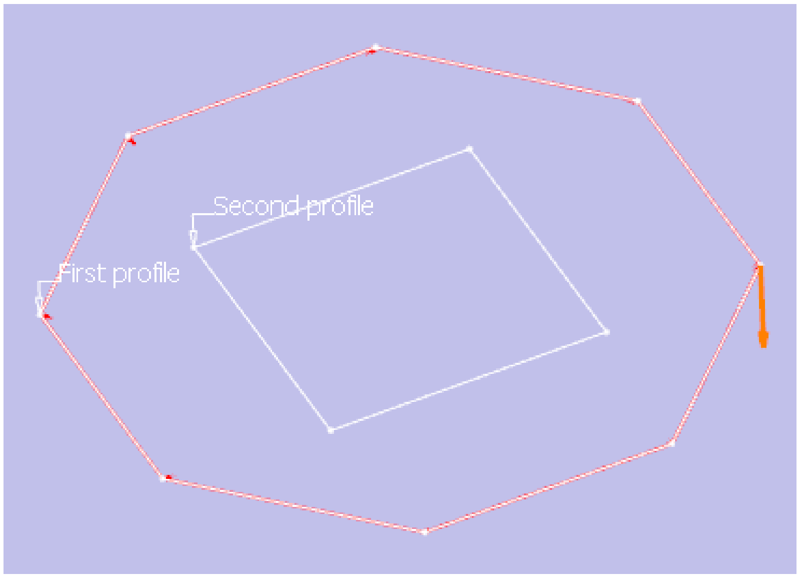
Select *OK*. The stamp appears as shown.



Select the Surface Stamp icon again.  The *Surface Stamp Definition* window appears again.

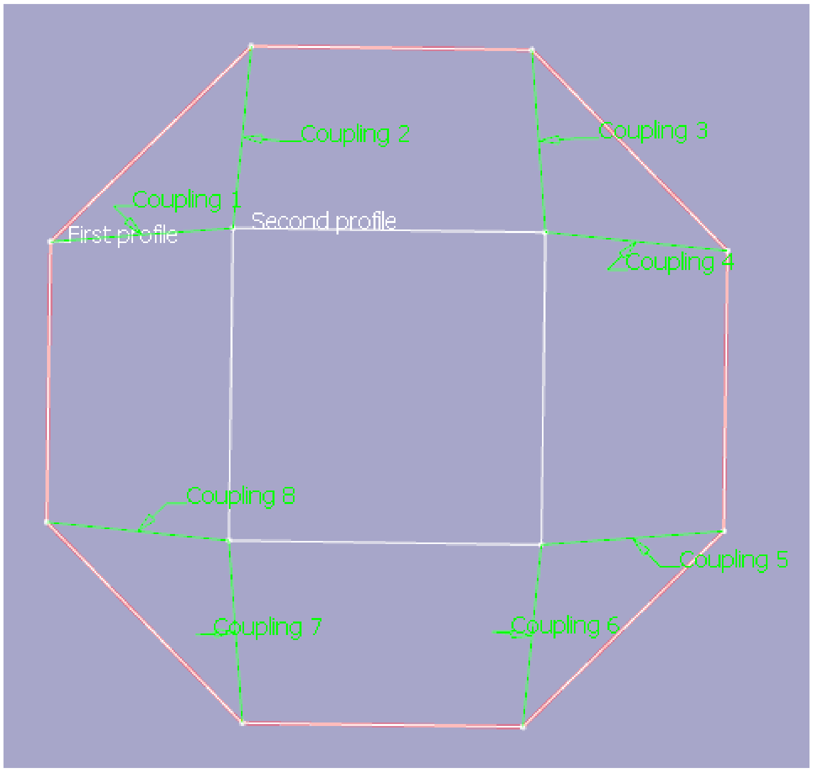
Switch the *Parameters choice* to be *Two Profiles*.

Select *Sketch.6* to define the *Profile* for the stamp and select *Sketch.7* to define the *Second profile* for the stamp.

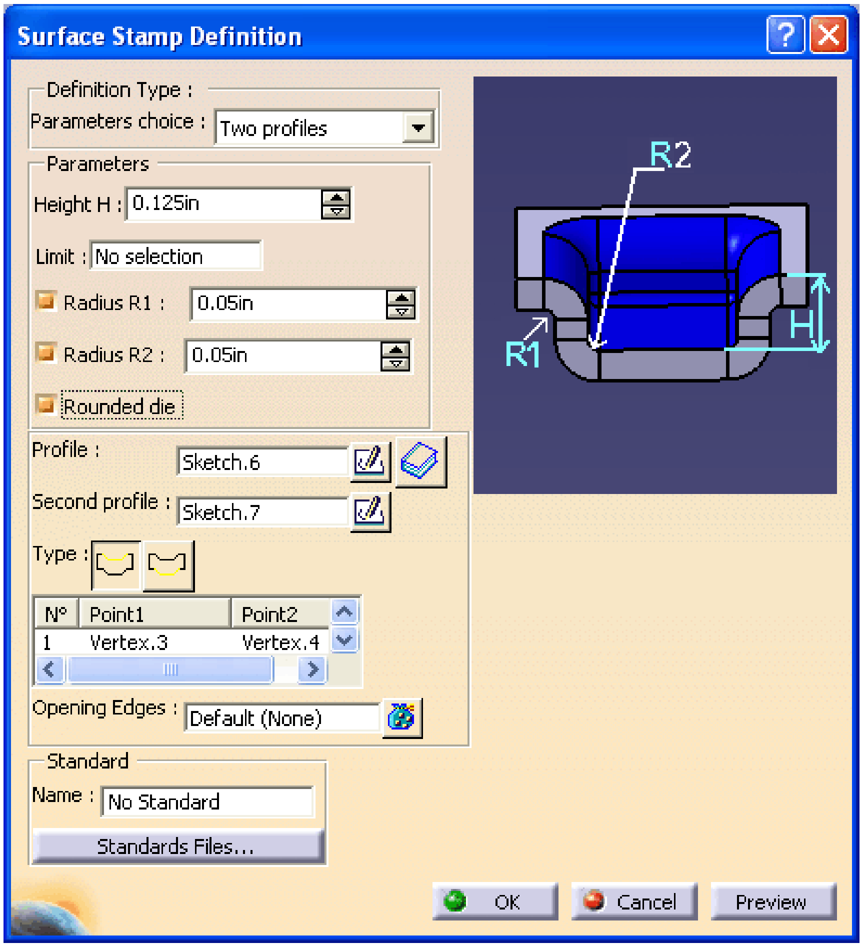


Notice the profiles are not of similar shapes. If you preview the stamp as it is now, you will get an error. Coupling curves will need to be created in order to specify how the vertices of the two profiles are supposed to match up.

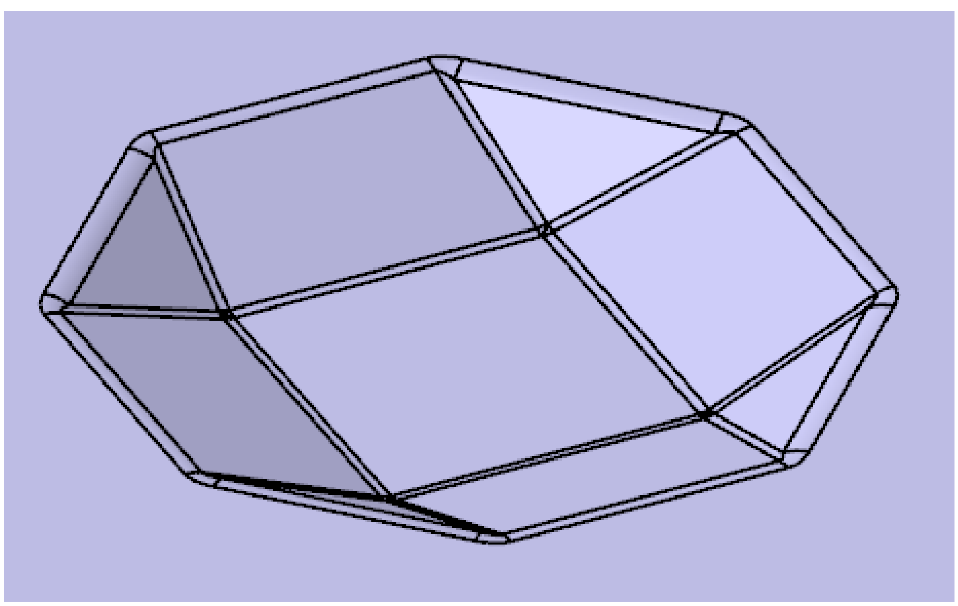
Define the coupling curves as shown. Simply select the two vertices that you want to match up and a coupling curve will be created. Make sure you select them all in the same order. For instance if you select a vertex on the outer profile first and then on the inner profile, define all of the coupling curves in that manner.



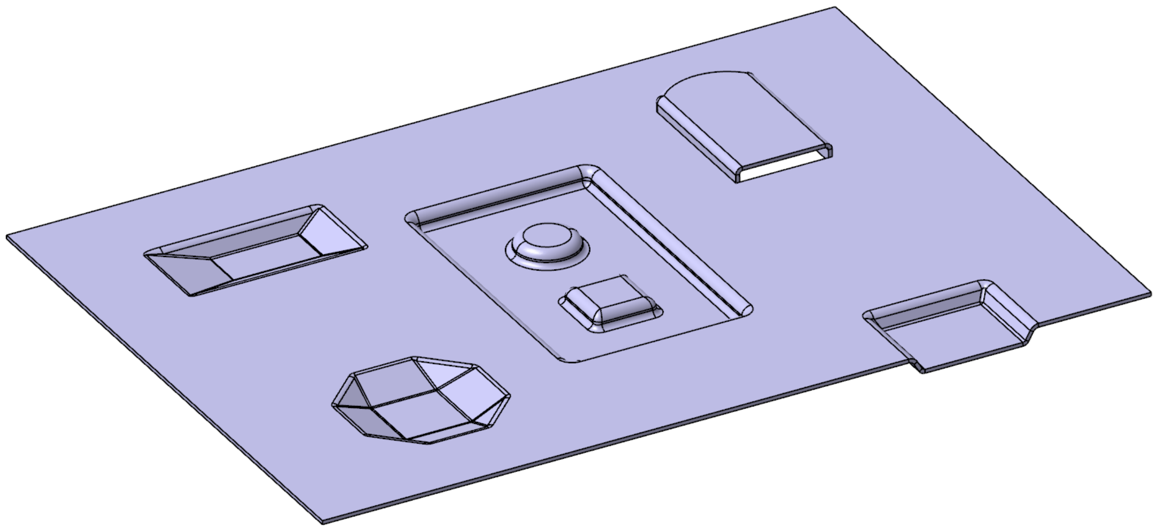
Set the options in the window as shown.



Select *OK*. The stamp appears as shown.



As you can see the surface stamp option allows a variety of different types of stamps to be created. This is probably the most versatile stamp option. Your final model should appear as shown.



Save and close the document.