

Plate Tutorial

This tutorial is designed to guide you through the steps in creating a sheet metal part. As you walk through this tutorial, procedures are designed based on your knowledge of earlier procedures. Because of this, it is very important that you carefully step through this guide, understanding everything along the way. We would like to cover a few key items before starting the step-by-step instruction of the tutorial. It is very important you completely understand these items:

Points The Part Interface is a simplified 2-dimensional CAD interface. To create parts, you must create the lines and arcs that compose the part. These lines and arcs (in most cases) are connected and created by first creating points. Using points to locate the features ensures the precision of the part. In the Assembly interface, PickPoints are used to locate parts and define the 3-dimensional points in the assembly.

The Assembly Interface The Assembly Interface is designed to allow you to assemble multiple parts together. Parts can also be made in the assembly, however it is usually quicker and more accurate (although not necessarily easier) to use one of the part interfaces to create parts.

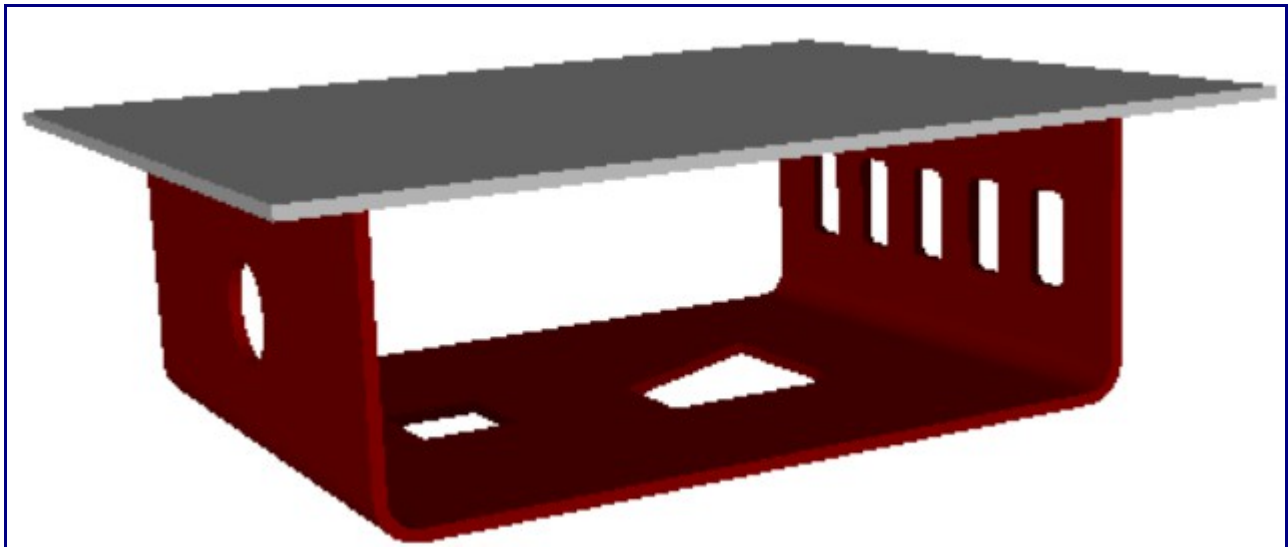
Sense of Direction The Part Interface is mainly a 2-dimensional interface. While you may view the part in full and partial 3D, editing and creating the features of the part take place in 2D. The Assembly however, is full 3D and uses PickPoints to locate features/parts. These pickpoints must be created prior to placing parts in the Assembly.

Free Select Free Select can be used to allow the user to click anywhere on the screen as if there was a point. Whenever Free Select is on, the display window is dimmed slightly. It is very difficult to place points with precision using Free Select mode, so make sure you take it off whenever you are trying to do something where precision matters.

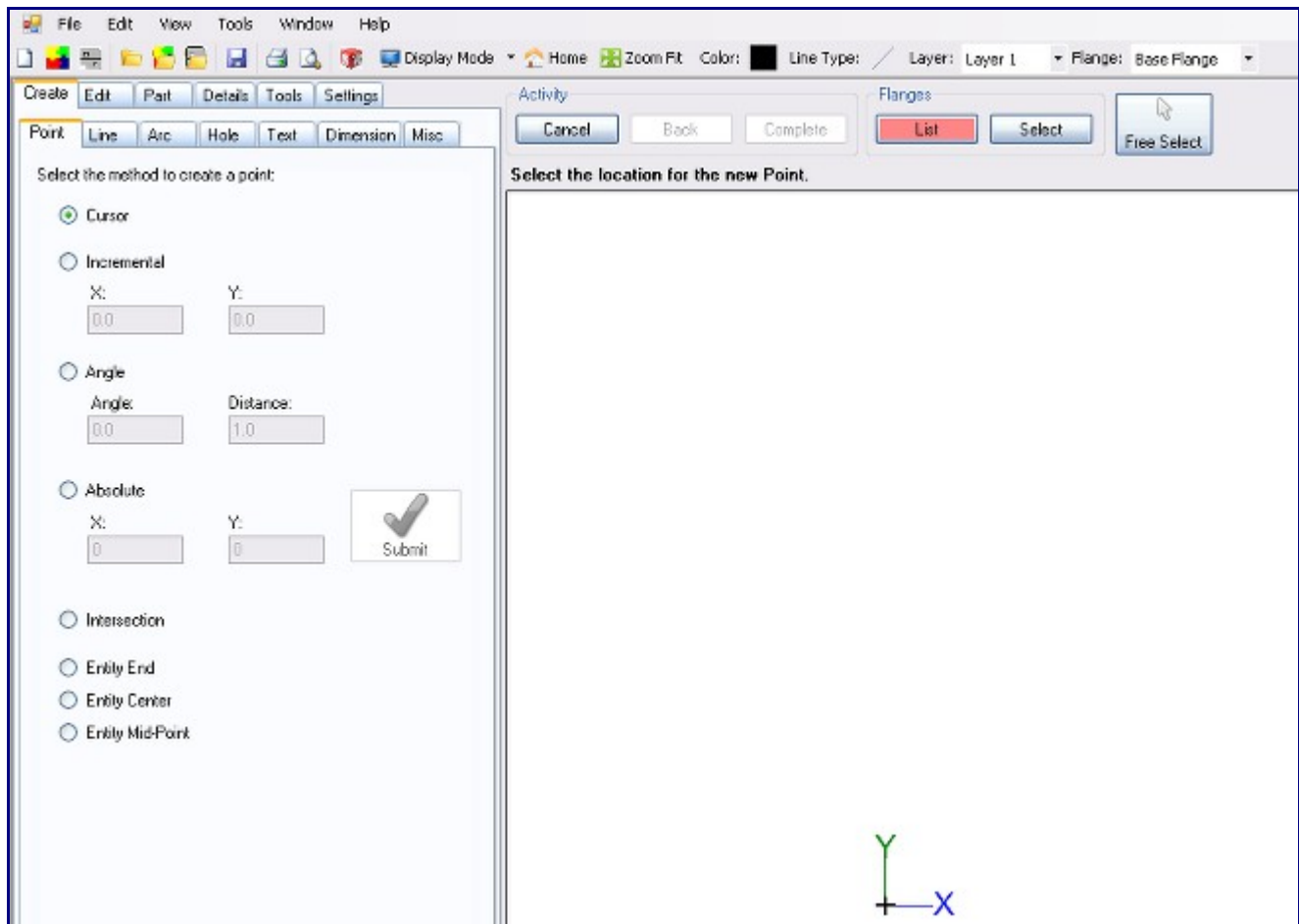
CAD vs Sheet Metal The Part Interface is basically divided into 2 areas, a simple CAD interface and sheet metal functions. The CAD functions allow you to create a part using points, lines, arcs, and other features for the part, much like other CAD software. These functions include trimming/extending lines, using tangent and intersection points, and placing dimensions. The sheet metal portion of the software allows you to convert features into a flange and add flanges together with bends.

Bending Radius When creating a bent part and setting up a material in the Plate Material Library, it is important to understand that the bending radius is the INSIDE radius of the bend. While some areas of the software may use the centerline dimensions, the bending radius of each bend on a sheet metal part is referring to the Inside Radius.

The following image shows the part we will be building in this tutorial.



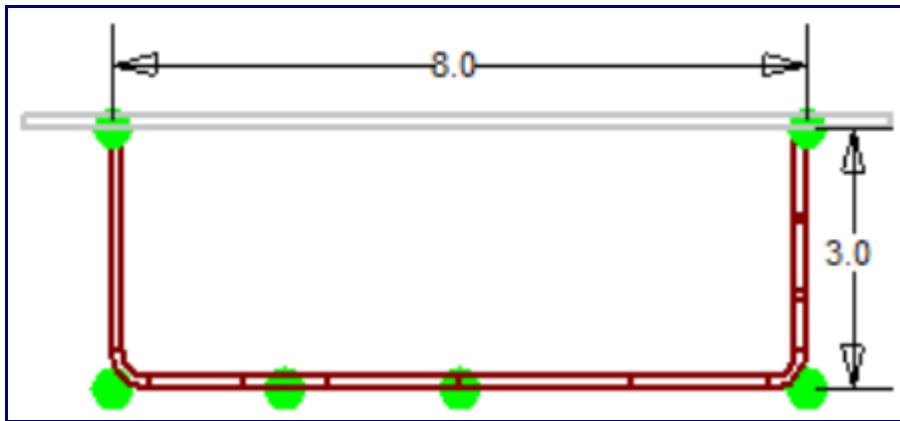
To begin, we must create a new part using the New Plate button on the main toolbar or by selecting File → New → Plate. The new plate interface will be displayed and can be broken down into two parts: The left side contains the features and value inputs, while the right side is where we see and select the features of the part. The large window on the right side of the screen is referred to as the Display Window in this tutorial.



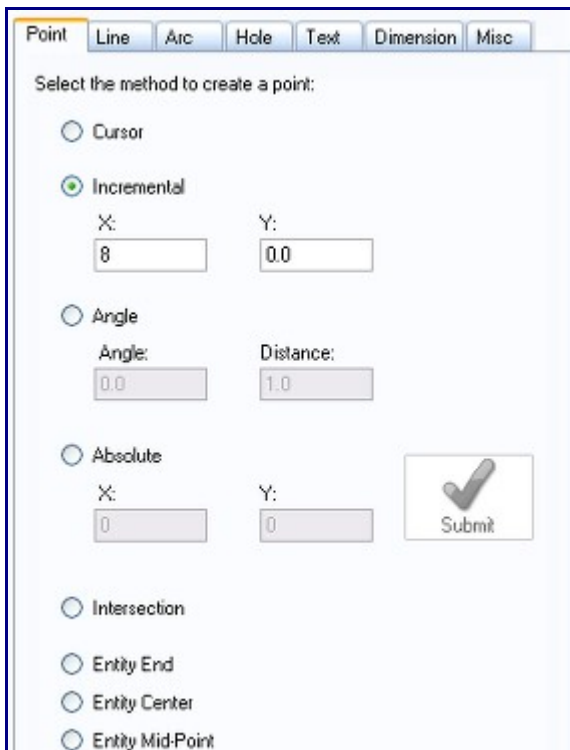
The first step to most any part is to create some points defining the part's base flange. All features of a part can be created under the Create tab (except for bends). Select the Create tab. The following tabs will be available:



The Points Panel will be displayed, showing the different methods you can use to create a point. You will also notice that the part already has a point created at the (0, 0) coordinate. We will use this point as the first point in our part. As you can see below, our first part is 8 inches wide, so we will create the base flange that wide. Select the Incremental point option.

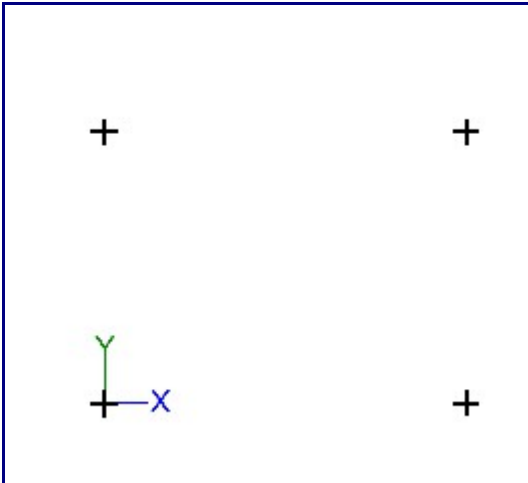


Enter a value of 8 in the X field as shown here. As you move the cursor over the part, you will notice a point 8 inches to the right. Now we need to select the base point to create the new point with the offset. Select the base point of the part and the software will create the new point 8 inches in the X-direction.

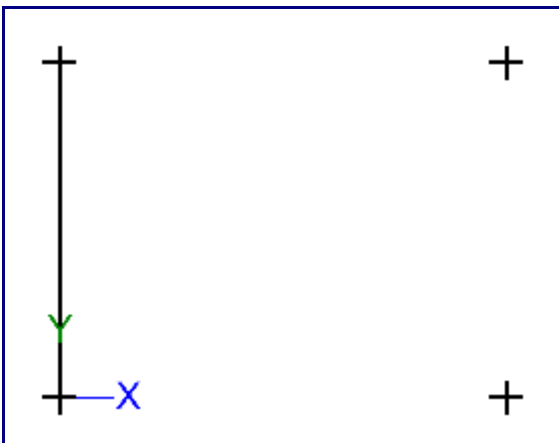




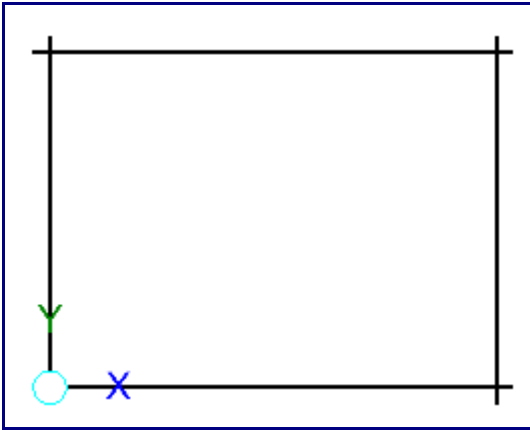
Next we will create a point 6 inches above this new point. Enter 0 in the X field and 6 in the Y field. This will create a point 6 inches above our cursor. Select the new point to create the third point of the flange. The next point we create will be 6 inches above the origin point. We already have this offset in the fields so we can simply click the origin point to create the fourth point.



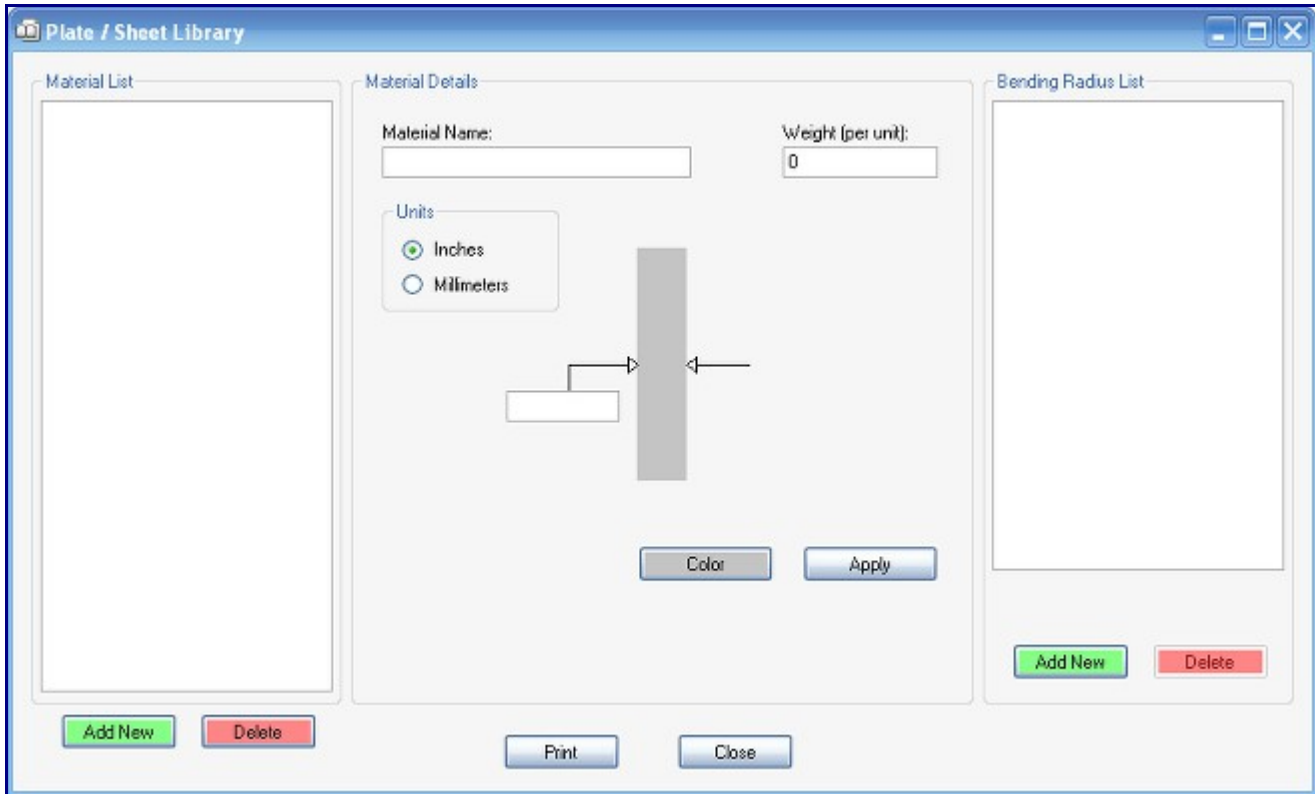
We now have the four points defining the outer edge of our base flange. We will now connect these points with four lines. Select the Create Tab (if it is not already selected). Select the Line Tab. Select the 2 Points option. Select the upper-left point for the first point of the line. Select the origin point as the second point of the line. You will now have a line created, connecting the 2 points you selected.



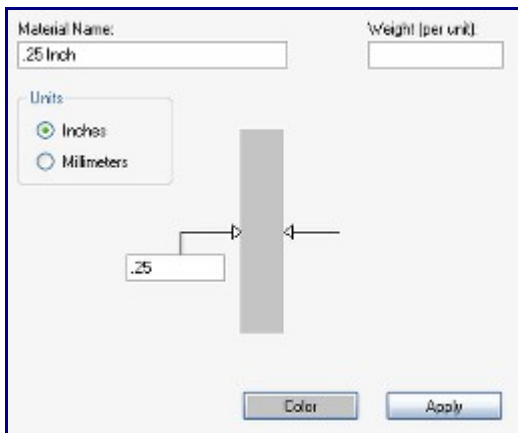
Click the Continuous checkbox next to the Two Points option. Click the top left point, then the top right point, then the bottom right point, and finally the bottom left point. There should be a rectangle drawn on the screen like in the picture below.



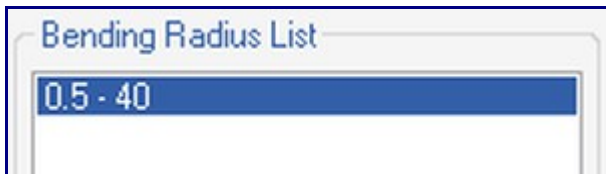
We now have completed our 8 inch by 6 inch base flange. Before we can add our second flange or view the part in anything other than 2D, we must define this flange. Select the Part Tab. In the Definition tab, click the Library button. The Plate / Sheet Library will open.



There are two Add New buttons. Click the left one. Enter “.25 Inch” in the Material Name field. Enter .25 in the thickness field (the field with the arrow pointing towards it in the picture below). Make sure Inches is selected in the Units area and click the Apply button.

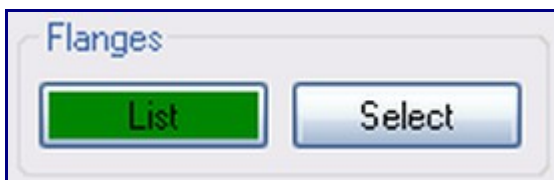
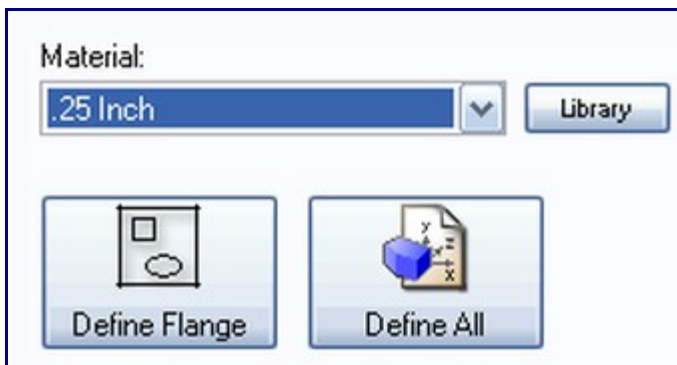


Click on .25 Inch in the Material List. Click the right Add New button. Enter .5 in the Inside Bending Radius field and click the OK button. Enter 40 for the K-Factor and click the OK button. Click the Apply button. Click the Close button to close the Plate / Sheet library.



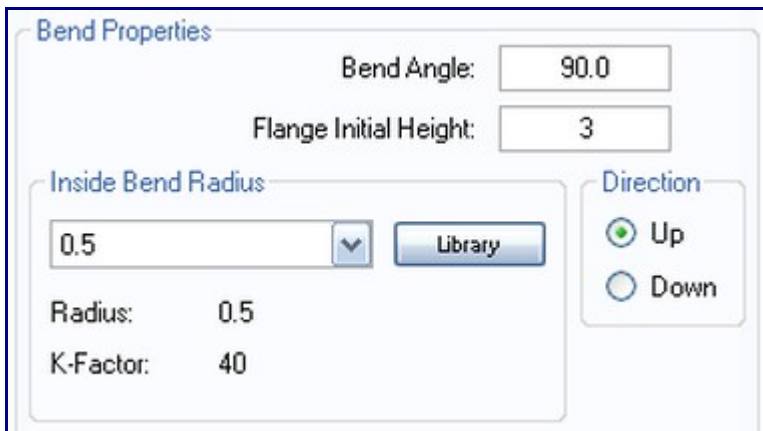
Go to the Part tab.

Select .25 Inch in the Material drop down menu. Click the Define Flange button. Click anywhere inside the box that you made in the display window. You should see the Flanges List button turn green, indicating that a flange has been defined successfully.



Go to the Bends sub-tab of the Part tab.

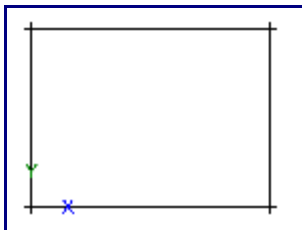
In the Bend Properties area, change the Flange Initial Height to 3. Make sure 0.5 is selected in the Inside Bend Radius field and Up is selected in the Direction area.



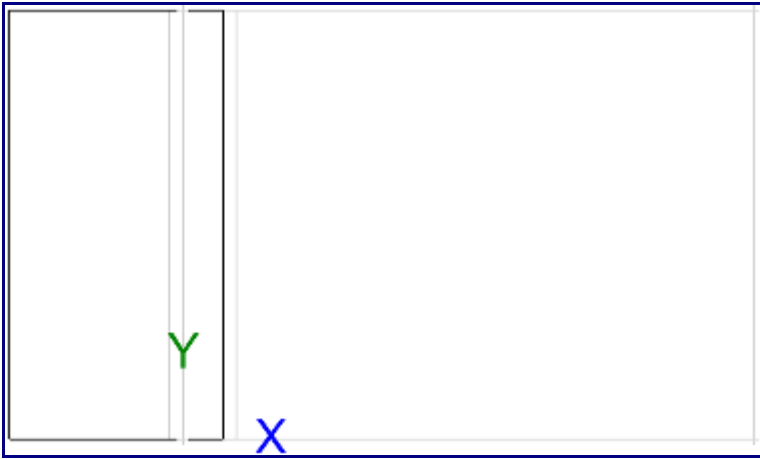
Change the Parent and Child Bend Location to Outside Apex.



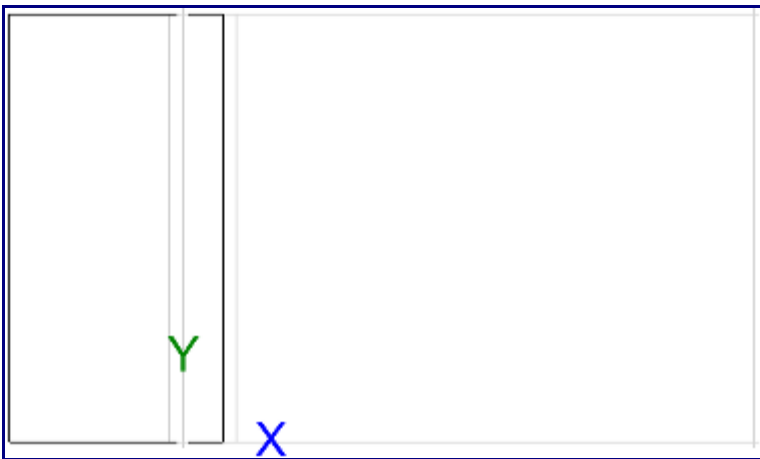
Click the New button. Click the line on the left side of the box you made. You will be prompted to name the flange. By default it will be named “Flange 2” which is good enough. Click the OK button. You should now see two new areas created on the part. The first area is the bend zone, an area between each flange that extends from the start of the bend to the end of the bend. The second area that was created is the second flange, which is built according to the Flange Initial Height we specified (3 inches in this case). Your screen should now look like the picture below.



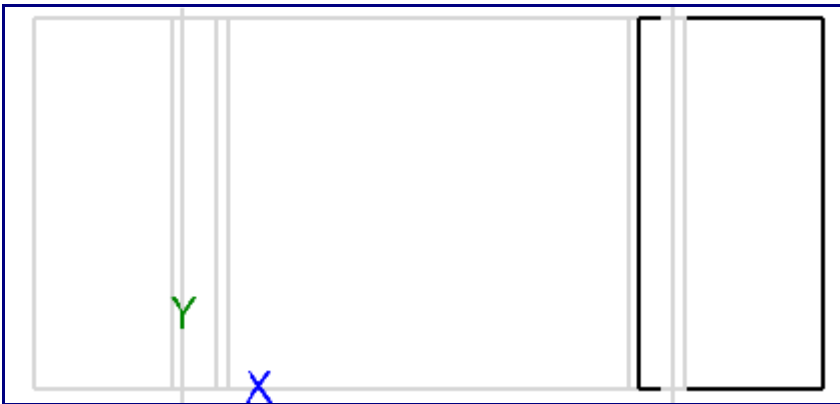
Note: The two flanges and bend zone will overlap, as we have selected Outside Apex for both sides of the bend. The bend zone extends from the tangent of each side. If we have selected tangent for both sides, the bend zone would not overlap.



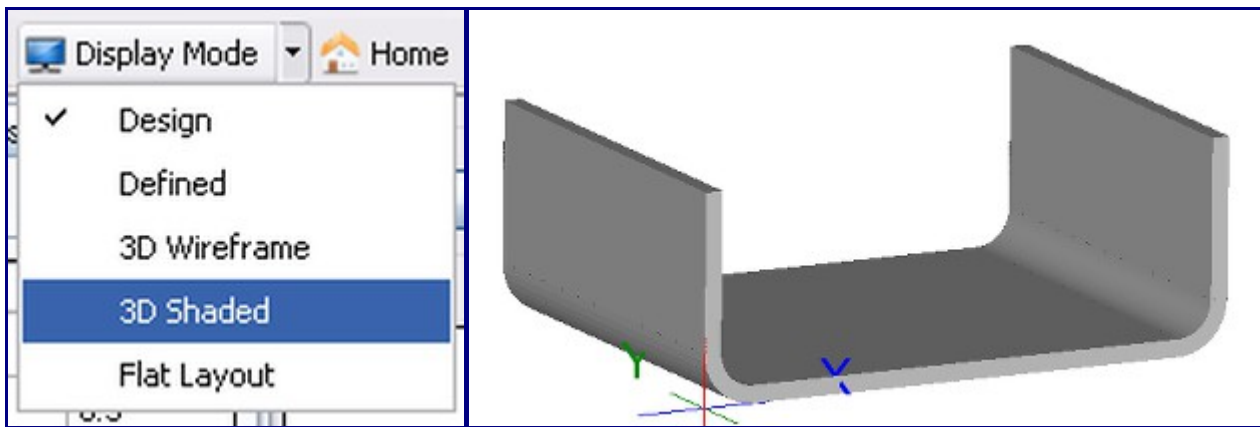
Click the New button. Click the line on the right side of the box in the display window.



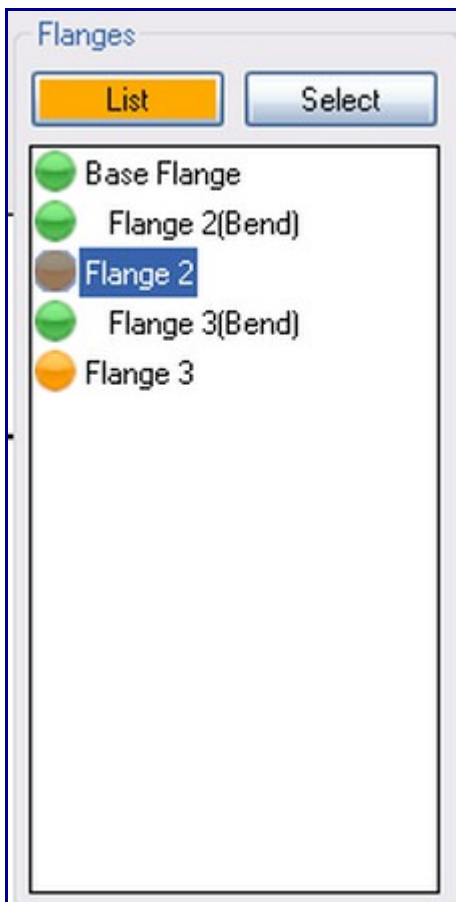
You will be prompted to name the flange, but “Flange 3” is good enough so click the OK button. Your part should now look like the picture below.



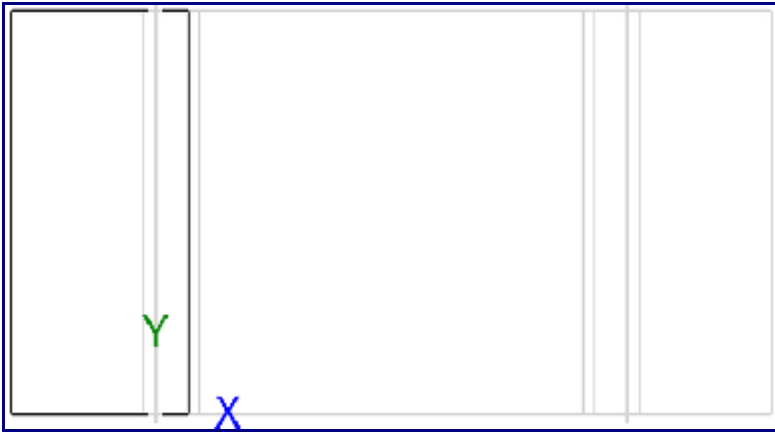
On the toolbar, click the arrow next to the Display Mode button. Select 3D Shaded. The part will now be displayed in full 3D. You can click and drag the right mouse button to rotate the display or click and hold both mouse buttons to pan. Use the scroll wheel to zoom in and out. Go back to the Display Mode menu and select Design.



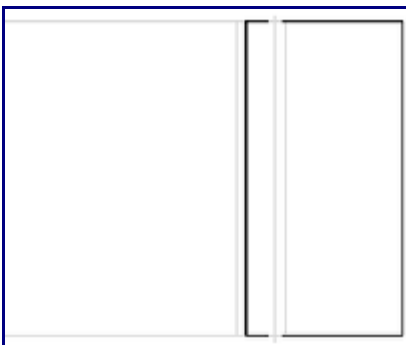
We need to define the two new flanges. Go to the Definition sub-tab within the Part tab. Click on the List button in the Flanges area. Double Click on Flange 2.



Click the Define Flange button and click on the line on the right side of Flange 2. See the picture below to make sure you are selecting the correct line.



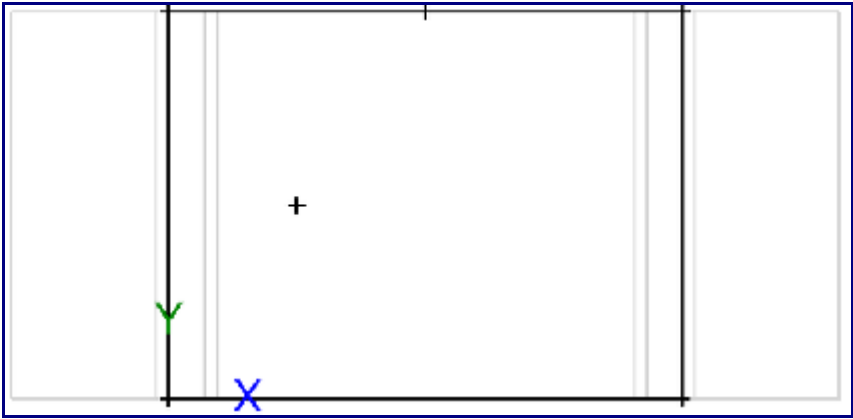
Use the List menu in the Flanges area to select Flange 3. Click the Define Flange button and click on the line that is on the left side of Flange 3.



Internal Features: Now that we have created the main structure of our part, we will begin creating the internal features of each flange. In order to create a feature on a flange, that flange must be the active flange. We will begin with the base flange. Use the Flange List to select the Base Flange item. The first flange we created should now be active (the other flanges should be gray). Referring to our part dimensions, we must create a square hole and a custom cutout in this flange. We will first create the square hole, however, we must create a point to place the hole with. Go to the Create tab and make sure you are in the Point sub-tab.

Select Incremental. Enter 2 in the X field and 3 in the Y field. Click the bottom left point of the base flange, which is also the location of the XY axis.

<input checked="" type="radio"/> Incremental	
X:	Y:
<input type="text" value="2"/>	<input type="text" value="3"/>

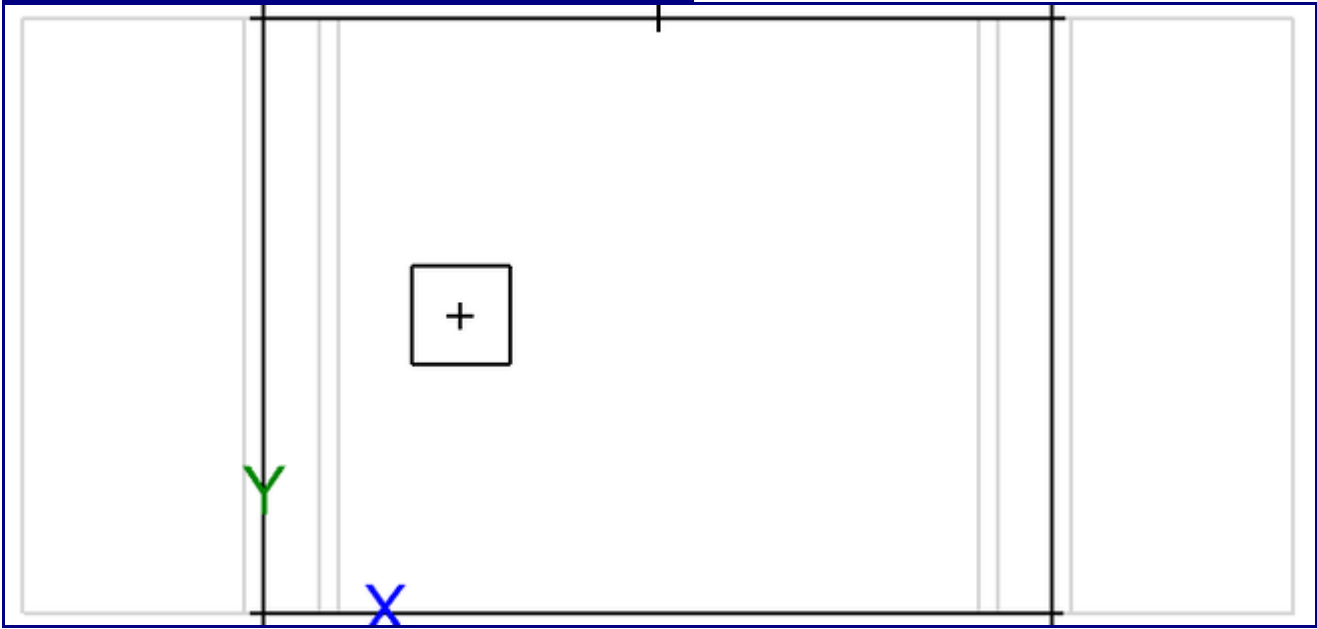


Go to the Hole sub-tab within the Create tab. Select Square in the Hole Shape area. Enter 1 in the Width field, 0 in the Angle field, and 0 in the Corner Radius field. Click the point that you just made to place the hole there.

Hole Dimensions

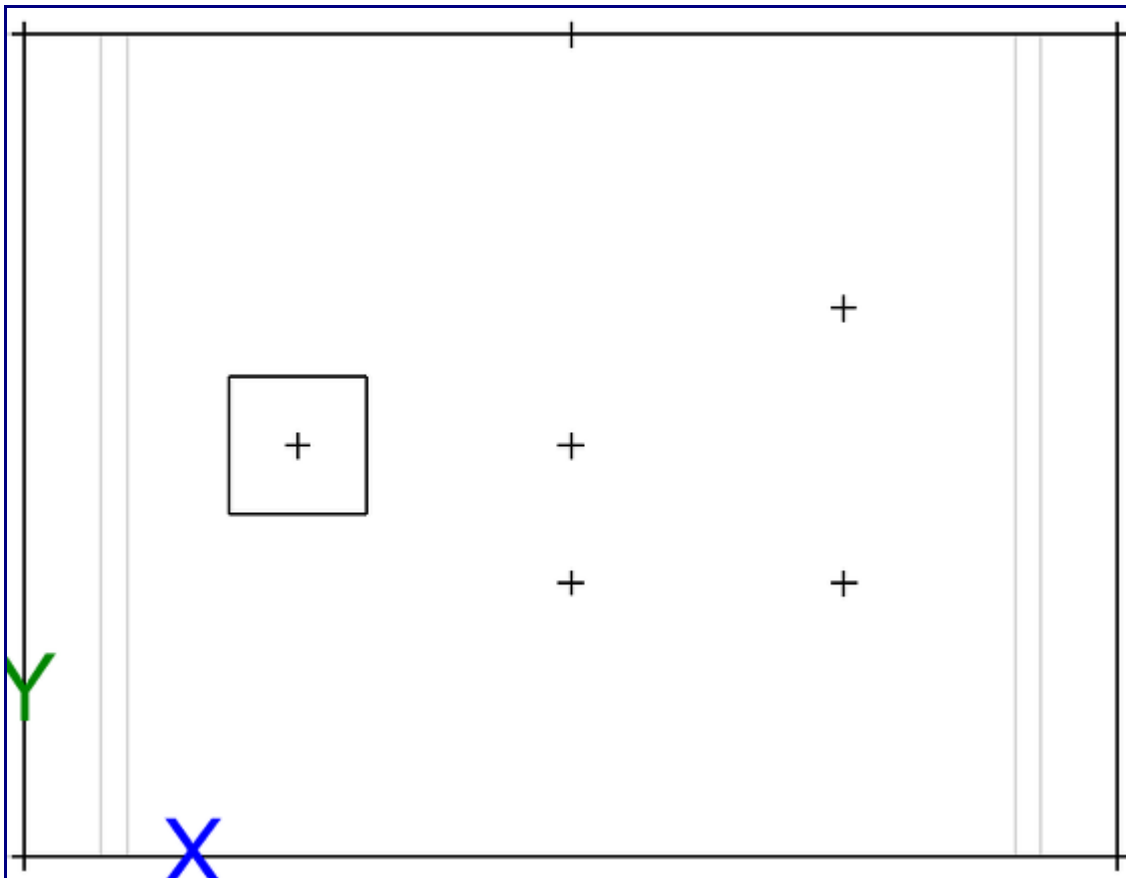
Width: Angle:

Corner Radius:

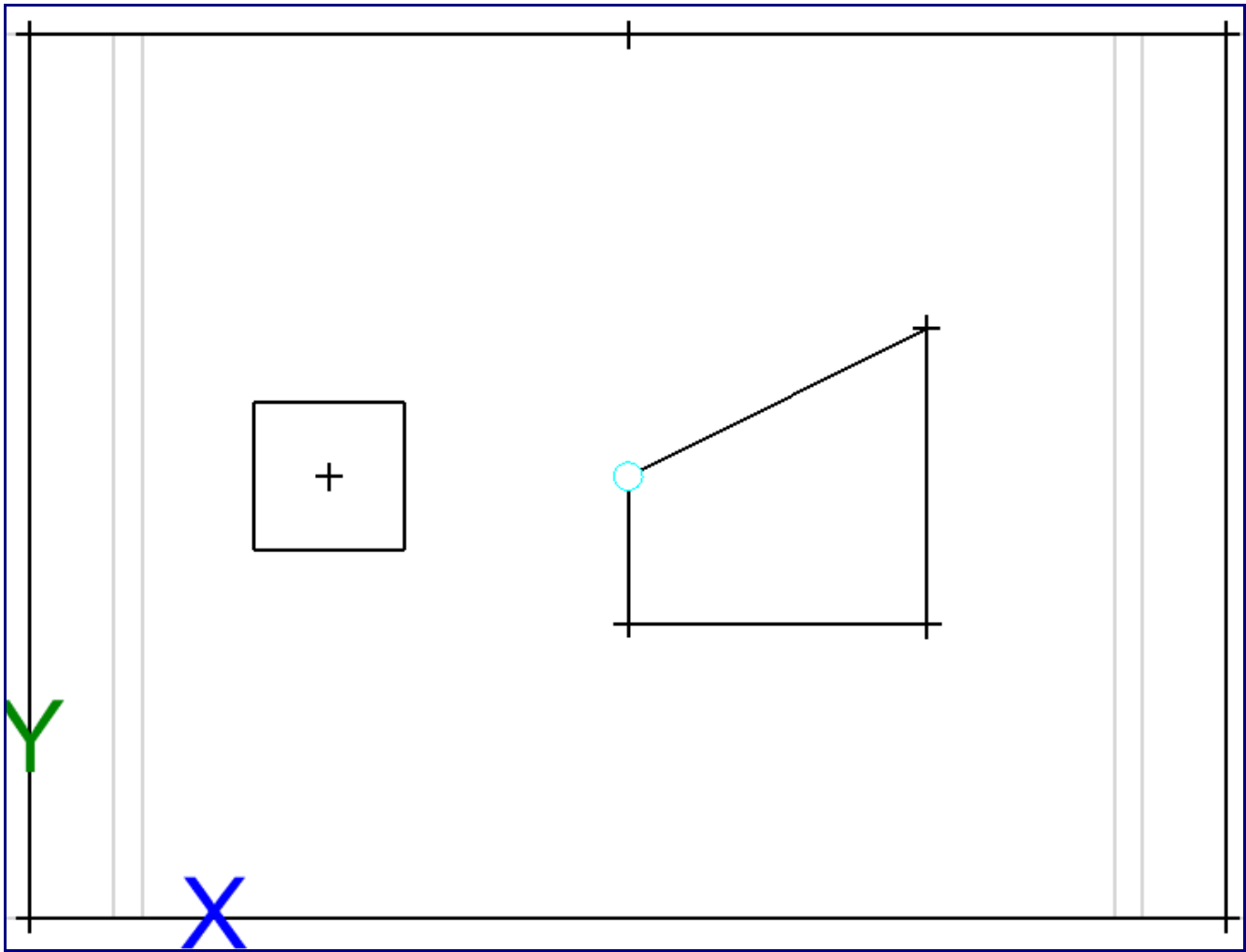


The upper-left corner of the cutout is 2 inches to the right of the center of our square hole. We will use the Angle method of creating a point to create this first point. Go to the Point sub-tab within the Create tab. Select the Angle option and enter Angle: 0, Distance: 2. Since we created the point for the square hole, we can use that as the reference point for the new point. Select the point at the center of the square hole to create a new point 2 inches to the right. Our next point is located 1 inch down from this new point. Change the values in the Angle fields to Angle: 270, Distance: 1. Select the last point we created to create the new point. You should now have 2 points created for the cutout. The next point is located 2 inches to the right of the last point we created. Change the values in the Angle fields to Angle:

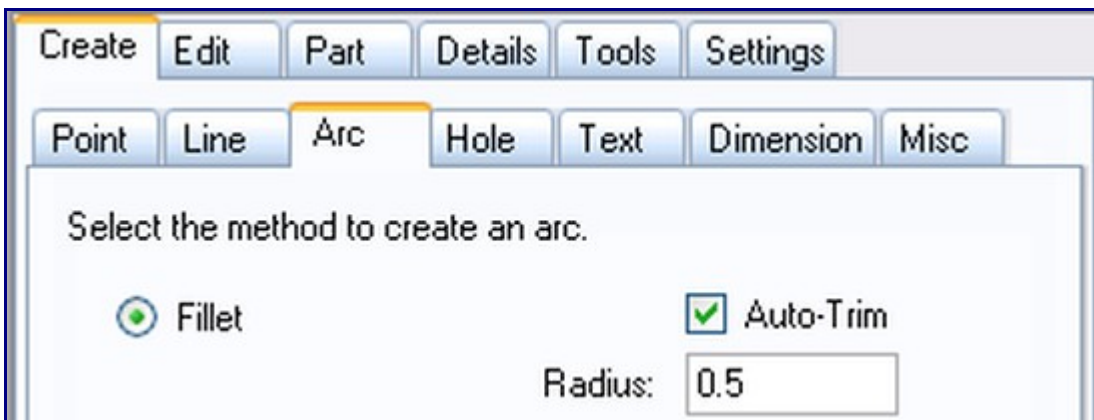
0, Distance: 2. Select the last point we created to create the new point. The last point is located 2 inches above the last point we created. Change the values in the Angle fields to Angle: 90, Distance 2. Select the last point we created to create the new point. We now have created the four points of our cutout. We can now connect these points with lines.



Go to the Line sub-tab within the Create tab. Select Two Points and check the Continuous checkbox. Select the upper-left point of our cutout. You will notice that the point turns into a blue circle. Select the point directly below this point to create the first line. Since Continuous mode is enabled, you may continue selecting points to create lines. Select the bottom right point of the cutout, followed by the top right point, and click the top left point to finish the shape. See the picture below.



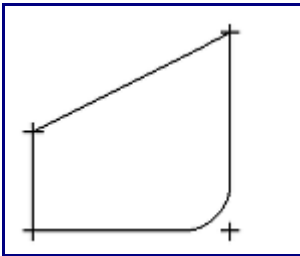
click the top left point to finish the shape. See the picture below. Once you create the last line, a line will still be attached to your cursor, as Continuous mode is still enabled. Select the Cancel button on the top of the screen to cancel creating lines. Go to the Arc tab. Select Fillet, change the Radius to 0.5 and check the Auto-Trim checkbox.



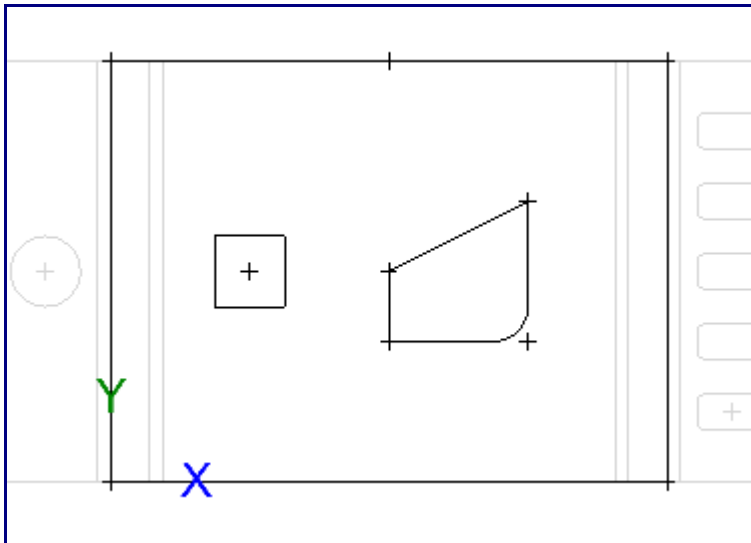
Click on the right line of the cutout, followed by the bottom line. This will create a fillet at the intersection of the two lines.



The cutout should now look like the picture below. We must now define both of the holes on the flange. To do so, all we need to do is define the flange like normal. The holes will automatically be defined along with the flange.



Go to the Definition tab within the Part tab. Click the Define Flange button and click the bottom line of the Base Flange.



We have completed the base flange of our part and will now move on to the left side flange. This flange has a single circular hole on it. Select the Flange List button and select Flange 2. Close the Flange List by selecting the List button again. The flange on the left of our part will now be the active flange. To create the circular hole, we will first need to create the center point. Select the Create tab. Select the Point sub-tab. Select the Entity Center option. The hole is centered on the flange, so we will use the Entity Center feature to locate a point on the center on the flange's edge line. Select the left-most vertical line of the flange. A point will be created at the center of this line. We must now create a point 1.5 inches from this edge. Select the Incremental point method and enter the values shown here:

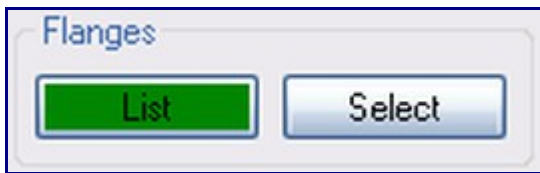
<input checked="" type="radio"/> Incremental	
X:	Y:
<input type="text" value="1.5"/>	<input type="text" value="0"/>

Select the new point we just created. A new point will be created to the right of it. Now that we have created the point for the hole, we can create the hole. We will use an arc for the hole. Select the Arc tab. Select the Values option and enable the Full Circle option. Enter a radius of 0.5 in the Radius field. Since we are creating a complete circle, the start and end angle fields are not available.

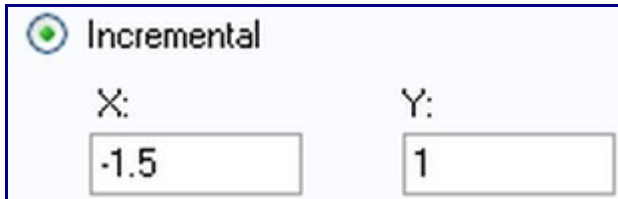
<input checked="" type="radio"/> Values	<input checked="" type="checkbox"/> Full Circle
Radius:	<input type="text" value="0.5"/>
Start Angle:	<input type="text" value="0"/>
End Angle:	<input type="text" value="360"/>

Select the point we created in the center of the flange to create the circle. Once again, we will need to re-define this flange in order to update the part. Select the Part tab. Select the Definition tab. Select the Define Flange button and then select the left-most vertical line of the flange. The flange and the circle hole will be defined.

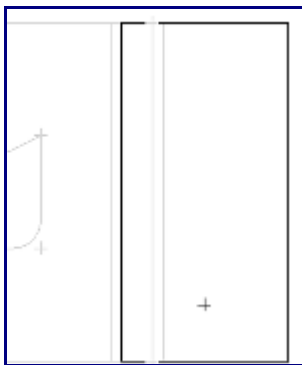
We must now create the holes in the final flange. Click the Select button next to the Flange area. We will use this method to select the flange we wish to work on. Select the far-right line on the right flange. This flange will become the active flange; we can now create the features.



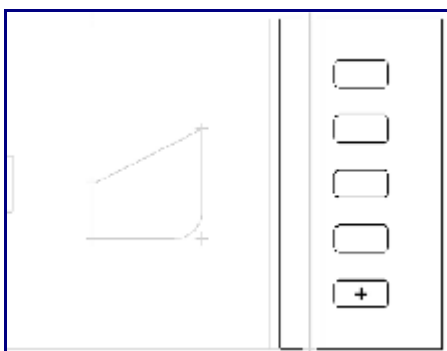
Select the Create tab followed by the Point tab. Select the Incremental option and enter the values shown here: We will be creating a point 1.5 inches from the edge of the part and 1 inch in. Note the negative sign in the X field.



To create the point, select the bottom right corner of the flange, this will place the point as shown in the picture below.

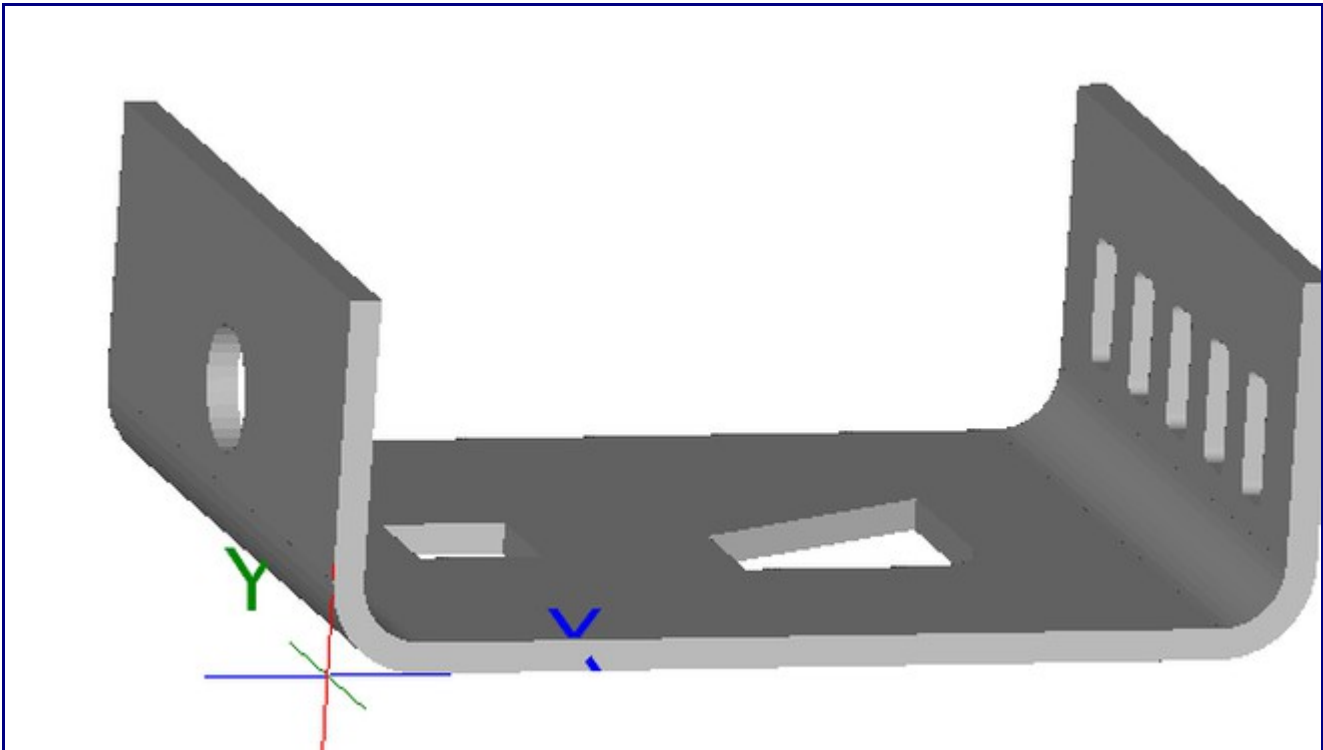


Select the Hole tab. Select the Rectangle shape and enter 0.5 for Width, 0.1 for Corner Radius, 1 for Length. Enter 0 in the Angle field. Click on the point that we just created to place a hole there. We now have the first hole of this flange. To create the remaining holes, we will use Move/Copy. Select the Edit tab and select the Move sub-tab. Ensure Cursor Select is selected under Selection. Uncheck the All box under Include Features. Next, check the Holes box to allow us to select only holes. Select the Move - Incremental option and enter 1 in the Vertical field. We also have to enable the Duplicate option, otherwise we will simply move the hole. We want to copy the hole and create 4 copies; this will give us the 5 holes needed on this flange. The holes will be evenly spaced 1 inch apart vertically. Check the Duplicate checkbox and enter 4 in the Number of Copies field. Select the rectangle hole we created earlier. Four copies of the hole will be created instantly. Your screen should look like the picture below.



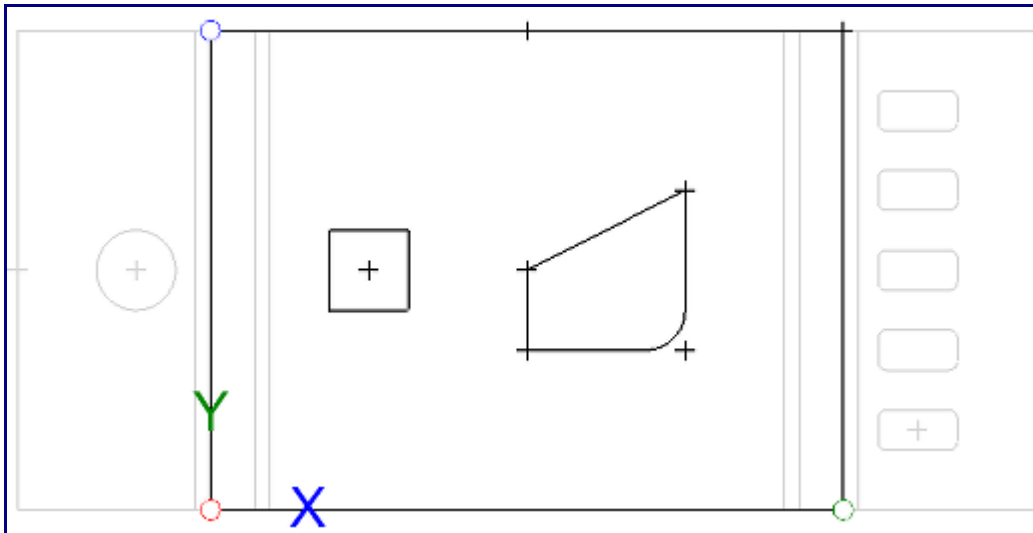
Go to the Definition tab inside the Part tab. Click the Define Flange button and click the line on the far

right side of Flange 3. In the Display Mode drop down menu in the toolbar, select 3D Shaded. Your part should look like the picture below. Use the Display Mode menu to switch back to Design view when you are finished viewing the part.



PickPoints: We will now create some PickPoints on the part for use in the assembly. As stated before, PickPoints are crucial to the placement of parts in the assembly. All creation and editing in the assembly is done with pickpoints. We will create some PickPoints on the part to allow us to assemble our parts. Be sure that the right flange is the active flange. If not, select it from the Flanges List before moving forward. Select the Create tab followed by the Misc tab. Inside the Misc tab, go to the PickPoint tab. We will create a pickpoint on the far edge of the part, on the bottom of the material. Ensure that the Depth is set to 0.0 and then select the lower-right corner of the right flange. The pickpoint will be created. Next we will create two pickpoints on the left flange of the part. Select the Flange List and select the left flange. Select the List button again to close the list. Select the lower-left corner of this flange to create the first pickpoint. Next, select the upper-left corner to create the second pickpoint. We have now created the three pickpoints we will later use in the assembly. Remember: the flange you are working on must be the active flange.

Send to Assembly: We will now send the completed part to an assembly. This will allow us to create the second part and assemble them together. Select the Part tab followed by the Assembly tab. Select New Assembly from the drop-down list. We must now select three points defining the base point and direction of the part. We also have to assign a depth to these points. We will define the points on the bottom of the material, so we will leave the Depth at 0.0. Click the Set button in the Base Point area. Select the lower-left corner, the lower-right corner, and then the upper-left corner of the base flange (in that order). The Send button should now be available. Make sure you are selecting the corners of the base flange and not part of the bend zones. You must take care when selecting the points. Once you have properly selected the three points, select the Send button. The image here shows the points to pick.



You will be prompted to give the part a name. Name the part however you wish and select the OK button. The part will be opened in the Assembly. You should see the part in the Master Part List and a preview of it to the right. To continue, we must first make the second part of this tutorial. To create a new part, open a new Part Interface by clicking on the New Plate icon in the main toolbar.



Select the Create tab followed by the Point button. Select the Incremental option and input values as shown here:

X:	Y:
<input type="text" value="1"/>	<input type="text" value="1"/>

Select the origin point to create the new point, 1 inch above and 1 inch to the right. Select the Absolute option and input the values shown here:

X:	Y:
<input type="text" value="9"/>	<input type="text" value="1"/>

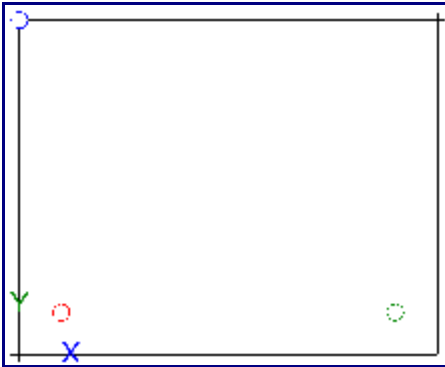
Click the Submit button to place the point 9 inches to the right and 1 inch above the origin point. The two points we just created will be used for positioning the part when we send it to the assembly. Now we must create a third point for the part itself. Select the Incremental option again and input these values:

X:	Y:
<input type="text" value="10"/>	<input type="text" value="8"/>

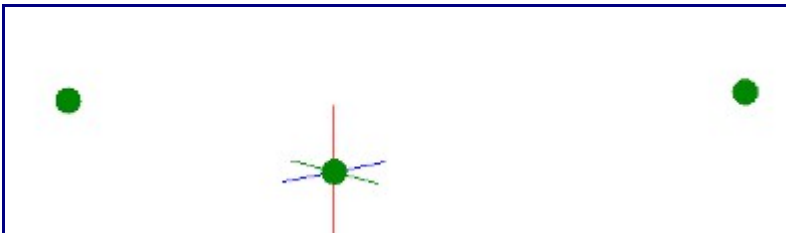
Select the origin point and the point will be created. We can now create the lines for this part. Select the Line tab. Select the Rectangle (2 Points) option. We are going to create a rectangle by specifying two opposite corners. Select the origin point as the first point of the rectangle. Select the point that is farthest to the top right. We now have created the rectangle part we need. However, we must first define it before moving forward. Select the Part tab. Select the Definition tab. Select the same material we

used for the previous part from the Material drop-down list. Select the Define Flange button. Click one of the four lines of the rectangle to define the part.

The part should now be defined. We can send this part to the assembly to assemble it with our other part we created. Select the Assembly tab. Select the Assembly drop-down list and select the assembly we already have opened. Now we must select the three anchor points for the part. We will use the two points we created earlier. Select the first point, the second point, and the upper-left corner of the part (in that order). Leave the Depth at 0 (zero) for all three of the points.

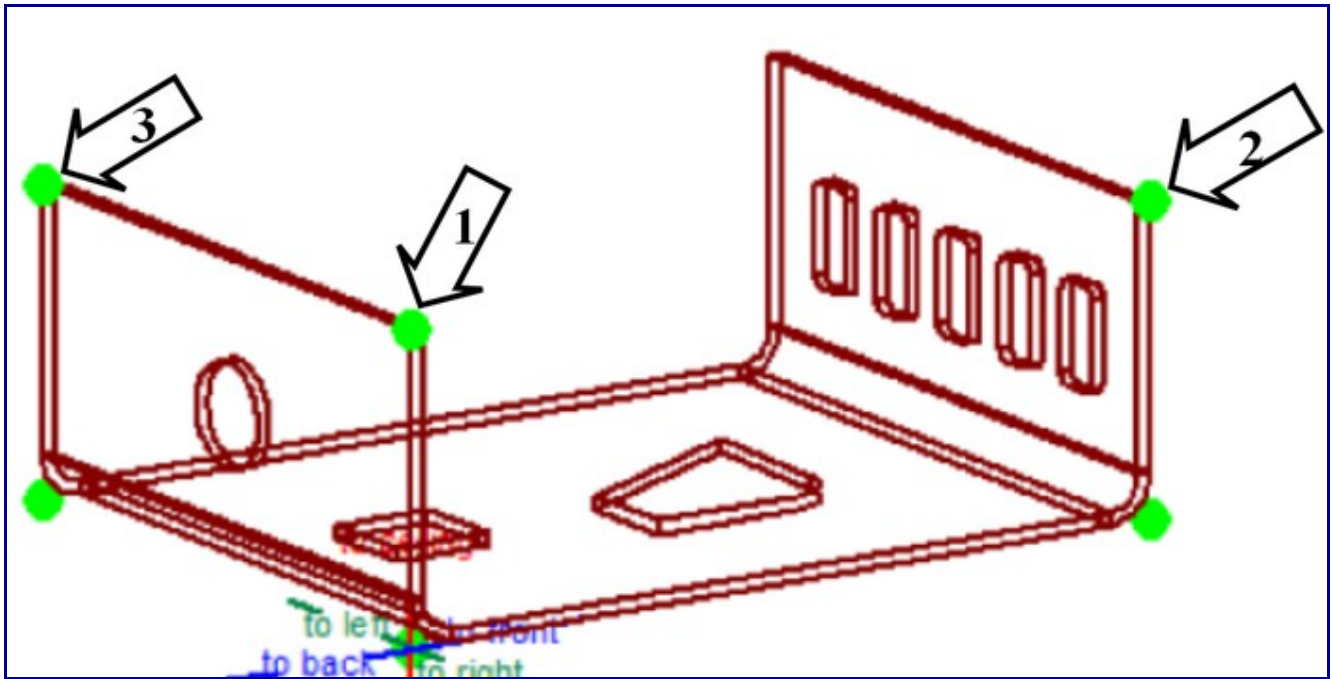


The Send button should now be available. Select the Send button. Leave the parts name as “Plate 2” and click the OK button in the Plate Name window. The part will then be opened in the assembly. We should now have two parts available in the assembly. Now that we have both parts created, we must create a few base pickpoints in the assembly. The pickpoints will be used to position our first part. Select the PickPoints Tab. Select the Reference Tab. Select the Set Reference button. We are going to create a pickpoint similar to using the Incremental method in the Plate Interface, but we select the reference point first, then enter the values for where the point is to be located. We already have one point created for us, it lies at the center of the origin or tri-star. This point is the (0, 0, 0) coordinate. Select this point. The point should turn red. Now we can enter the values for the new point. Enter 8 in the Front field. Click the Apply button, followed by the Clear Values button. Enter 6 in the Left field and click the Apply button.

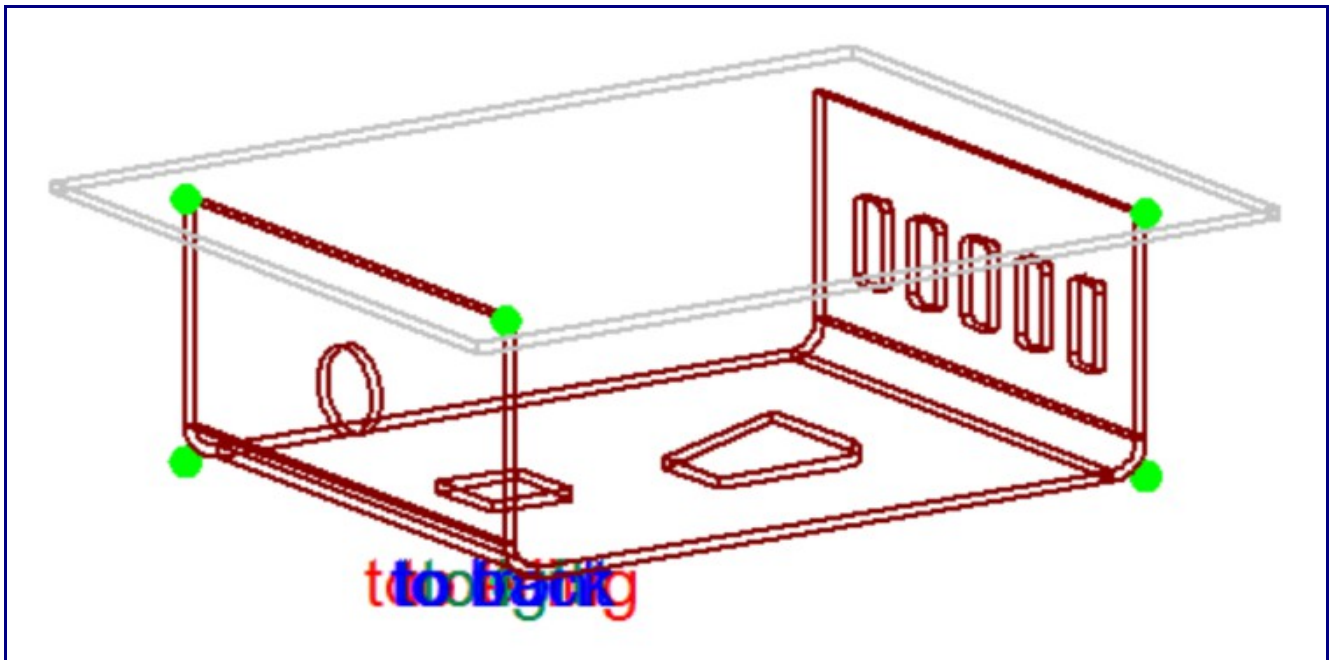


Select the Main Tab. Select “Plate 1” (the 3-flange part). Select the Paste Part button. The part is now ready to be pasted into the assembly. To add the part, we must select three anchor points in the design area. These points will coincide with the three points we used when transferring the part to the assembly. Select the origin point as the base point for the part. The point should turn red. Select the first point we created (in the to front direction) as the second anchor point. The point should turn red. Select the second point we created (in the to left direction) as the final anchor point. The part will now be displayed in the design area. You should notice the PickPoints we created with the part are displayed with the part. We will use these points to attach the next part.

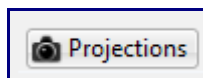
Select “Plate 2” from the Master Parts List (part with no bends). Select the Paste Part button. Once again we need to select three points in the design area to paste the part. Select the three pickpoints that we created with the first part as shown below in the specified order.



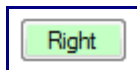
Once you select the third point, the part will be added to the assembly. Your assembly should now look similar to the picture below.



Select the Projection button at the top of the design area.

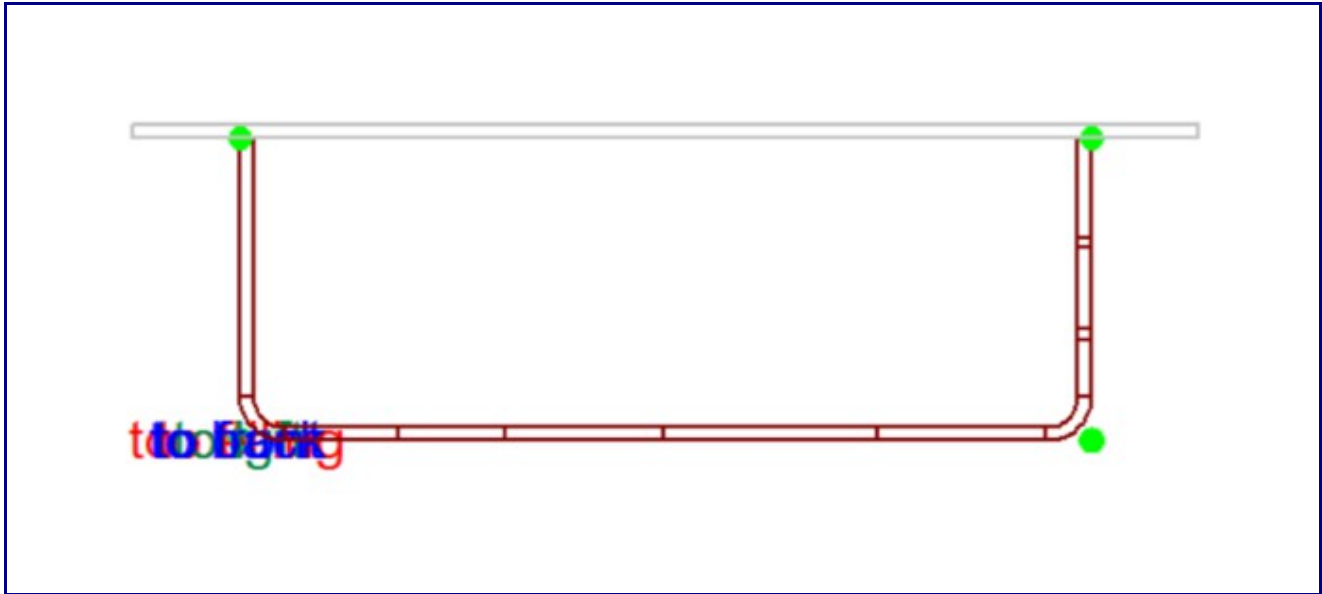


Select the Right button.

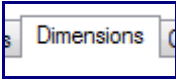


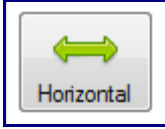
The view will re-orientate itself so that you are looking at the assembly from the right side. Your

assembly should look similar to the picture below.

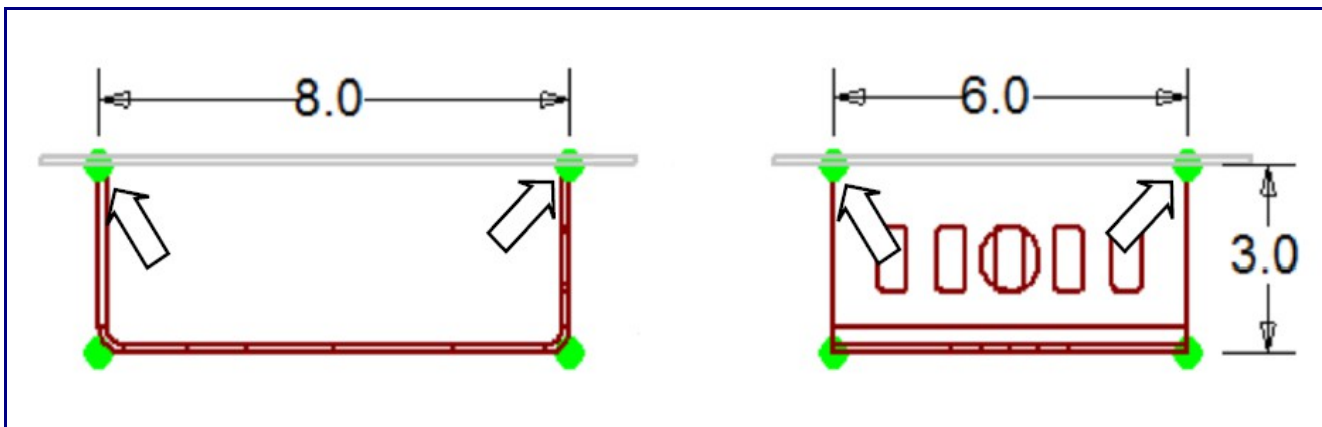


Dimensions: We will now add several dimensions to the assembly to verify the overall dimensions of the parts.

Click the Dimension tab. 

Select the Horizontal button. 

We will create a dimension measuring the overall width of the first part. Select the two pickpoints shown below-left, you will then need to select a location to create the dimension. Select the two points shown below-right to create a dimension measuring the width of the part.



We will now add a dimension to measure the height of the first part we created.



Select the Vertical button.

Select the bottom and top point of the right side of the flange to create the vertical dimension shown above.

We have now concluded this tutorial. You may experiment more with this part if you wish. For further information, please use the Help menu in the software or visit the rest of the wiki pages here.