
Curve from Sketcher - I

This webtask will familiarize you with the methods of evaluating shapes and creating 2D curves from a 3D object. Our goal is not to create an accurate model but know the tools and the methods to use them inside thinkdesign. In this task, you will use curve creation and curve editing commands to create the base for the framework on which we'll build our 3D geometry. Please follow the steps below...

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1. Step 1: Let's start sketching

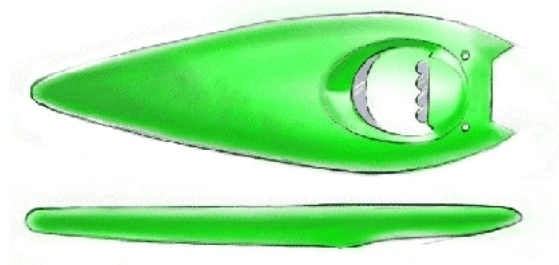
Let's open a new model file to insert an image of a bottle opener and start the sketcher task on it. We'll create planar entities using mainly lines, arcs and circles.

Note:

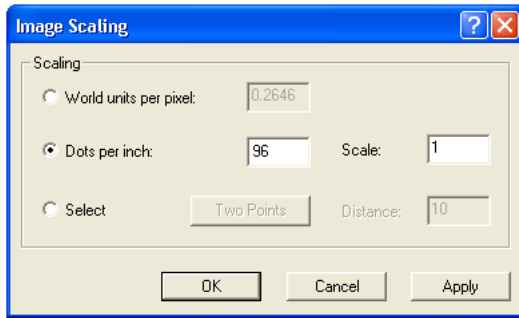
With a double click on the exe file you can run the webtraining session. thinkdesign will open automatically in the correct model.

If you're prompted to open a file, you can find it in the C:\MyTraining path.

- Suggest to use Industrial Design GUI.
- Click on the **Insert Image** button to bring in a concept sketch of our bottle opener.
- The image file resides in the files folder inside the task folder of the downloaded webtask



Select **Scale Image** from the Image Toolbar and select the image in the graphic area. In the pop-up dialog box enter the values as shown in the image below.



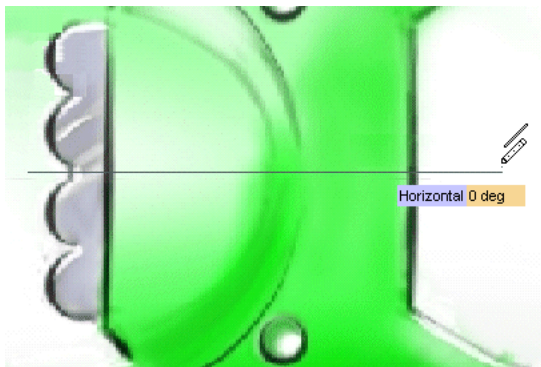
Consider this image as a sketch from an artist or a concept designer. You will find that the shape sketched is not perfectly symmetrical about the reference axes. Moreover there are other defects that are typical of hand sketches. This is not a problem, as our task will be to define and obtain an accurate sketch based on this rough sketch while retaining the original creative concept of the artist. Let's begin with the task.

First, let's draw a few construction lines that we will use for reference. The image looks symmetric about the longitudinal axis. So, let's start here.

- Zoom in to the top end of the opener's head so that our center line will be more accurate.
- Select **Insert** → **Drafting** → **Line** → **2 Points** or select the command from the drafting toolbar and draw a horizontal line dividing the image at the center, as shown below.

Note:

Don't be too concerned if your construction lines or curve dimensions are different from those in images below. You are working on a hand sketch and such variations are bound to occur.



Now, let us extend the line length so that it will cover the entire model. If you double click toward the end of the line, both the angle and length dialog box of the line will be highlighted. We want the line to remain horizontal, so follow the steps below.

- To extend the line, keeping the angle locked, double click the line at its midpoint. Selection in this area will allow the line to extend in a linear fashion only.
- Extend the line till it overlaps the entire sketch as shown in the image below.

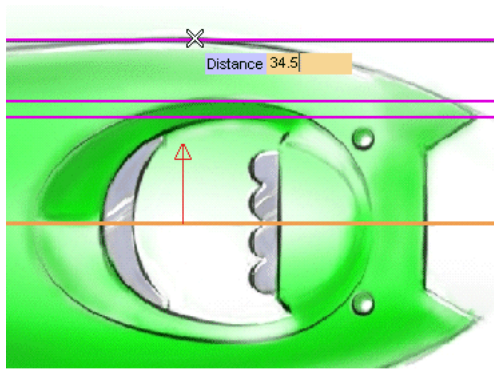


Note:

Same result we can obtain by **Update Parameters**.

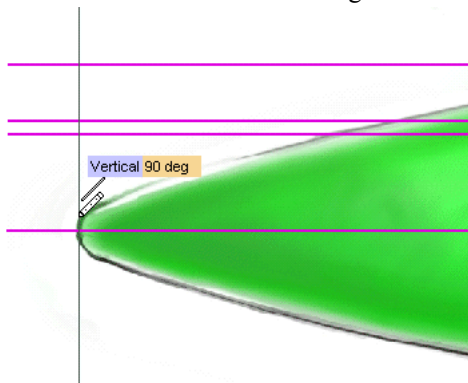
Since this part is symmetric about the line we just sketched, we can create half of the model and then mirror it to save us some time. Let's sketch some key horizontal lines.

- Go to **Insert** → **Drafting** → **Line** → **Parallel**. Select the center horizontal axis as reference line.
- Set Type option to Copy Length.
- Draw parallel lines at the very top of the model and another two lines at the upper point of inner two conical curves as shown. If you input the dimensions in the mini-dialog and hit the lines will be entered aligned.



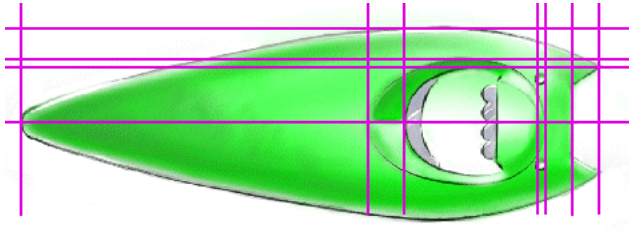
Now, draw a Vertical Line at the left most point for reference.

- Go to **Insert** → **Drafting** → **Line** → **Angular** command. Set the angle to Vertical 90 deg.
- Zoom in to left side of the image and choose a point at the "tail end" of our part to place the line.



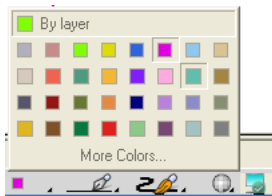
Now, let's draw some vertical reference lines parallel to the current line, just as we did for the horizontal axis, a few minutes ago.

- Use **Insert** → **Drafting** → **Line** → **Parallel** command and set Copy Length option.
- Select the left vertical line as the reference line and create parallel lines that lie on the major features of the sketch. Use the image below as reference.



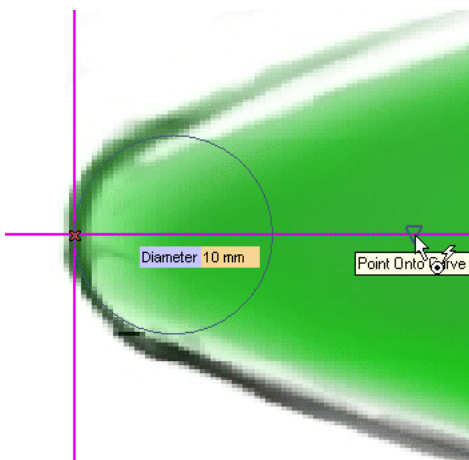
Let's change our default color and line width. This will help distinguish the model curves we're about to create from the reference curves that we've already created.

- Hit the **Esc** key to make sure that nothing is selected.
- From the attributes toolbar in the lower right corner of the thinkdesign window, change the current color to #5
- Then, from the same toolbar, set the current line width to #2.



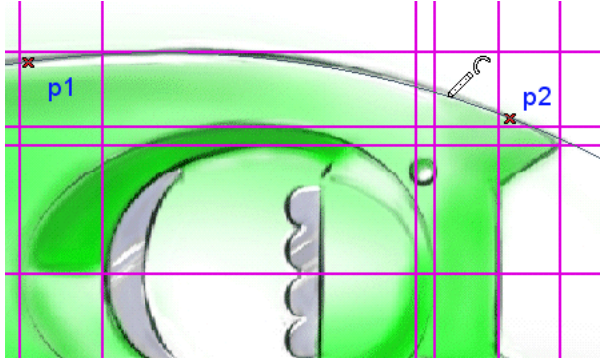
Let's start creating curves upon which we can build our surfaces. Having the reference lines to help us -- we will first build curves on the tail end of our Bottle opener sketch.

- Go to **Insert** → **Drafting** → **Circle and Arc** → **Radius**.
- Set the Given mode to Point.
- Select the intersection point of Horizontal axis and left side vertical line.
- Judge the Diameter of the circle. It should cover the tail end of the bottle opener sketch. Change the value if required and size of the circle according to your judgment. Use the image below as reference.
- Set the Snap option to **Point on Curve Snap** and select any point on the Axis as reference Given Point to draw this circle.



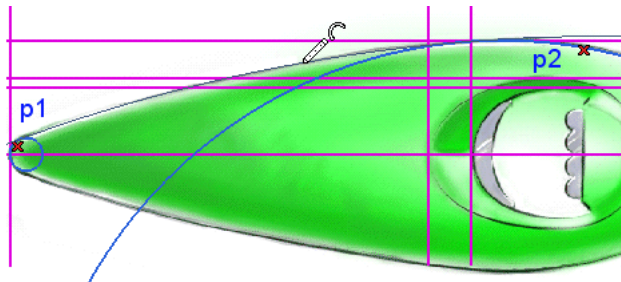
Let's move to the right side. Zoom in at the right top corner. We'll draw two circles which will encompass the outer boundary of the model. The first circle will be in the upper right area of the sketch.

- Go to **Insert** → **Drafting** → **Circle and Arc** → **3 Points** command.
- Select two points to start the 3-Point circle; one is very close to top edge of the model and other one is very close to right end of model. See image below for reference.
- Select the third point in such a way that the circle created will make a partial boundary for the model.



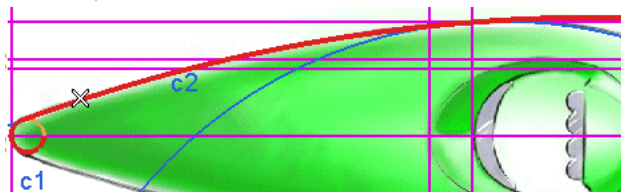
Now, we will bridge the gap between these two circles we've drawn by creating another circle tangent to these two circles. Finally we will trim each of them to get the final outside shape of our part.

- Go to **Three-Point Circle** command again.
- Set the Point snap option to **Tools** → **Snap** → **Tangent**. Select the first point at the top of the smaller circle. The exact placement isn't important. The tangent snap gives us a lot of flexibility.
- Again, click on the **Tangent Snap** Button. Select the second point on the top edge of the big circle we just created.
- Lastly, to complete the circle, pick the third point at the top of the mid-body of our part. Reference the image below.



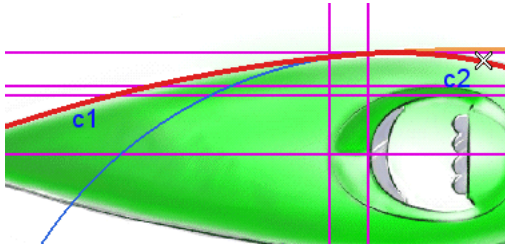
Now, we need to Trim/Extend the edges to get the correct output.

- Start the **Edit** → **Trim/Extend Curves** command.
- First; select the small circle on the left and then the very large circle which is tangent to it.

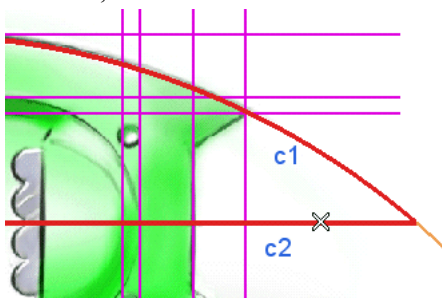


- Repeat the **Trim/Extend Curves** command.

- First select the very large 'tangent' circle, in the area bridging the two circles.
- Next, select the right side circle. It will delete the extended part of right side circle.

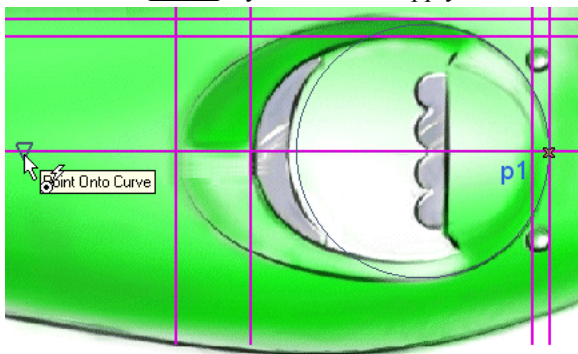


- **Edit** \Rightarrow **Smart Delete** the lower half circle of the small, left side circle.
- Then, **Trim/Extend Curves** the axis line with the right circle.



This completes the outer profile of our model. Similarly, let's move ahead and draw inner two conical shapes for the opening in our bottle opener. We'll draw the outer conic curve first and then the inner one.

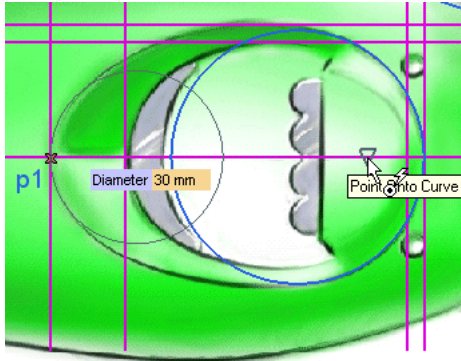
- Start the **Insert** \Rightarrow **Drafting** \Rightarrow **Circle and Arc** \Rightarrow **Radius** command. Set the Given option to Point.
- For the first point, select the intersection of the axis line and the vertical line on the right side of the conical shape as shown below.
- Set the Selection list Option to Diameter. Enter a diameter size that fits the right side of the oval shape.
- For the second point, use the **Point on Curve Snap** command and select a point on the axis line.
- Press the key or click the Apply button to create the circle.



Draw one more circle on the left side of the oval shape in a similar way. Change the Diameter value to match the shape in the sketch. Review the image below as a reference.

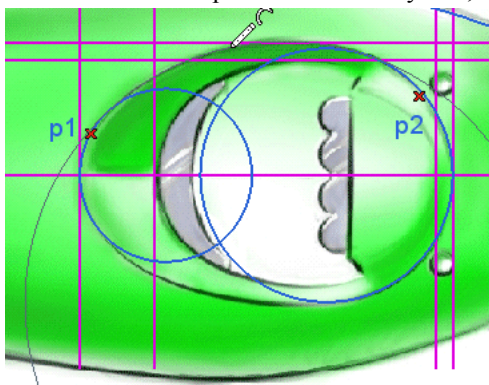
- Select the intersection point of the axis and vertical line on the left side of the oval shape.

- Change the Diameter value to match the shape in the sketch.
- For the second point, use the **Point on Curve Snap** command and select a point on the axis line.
- Press the key or click the Apply button to create the circle.



Now, we will draw a circle which will bridge these two circles.

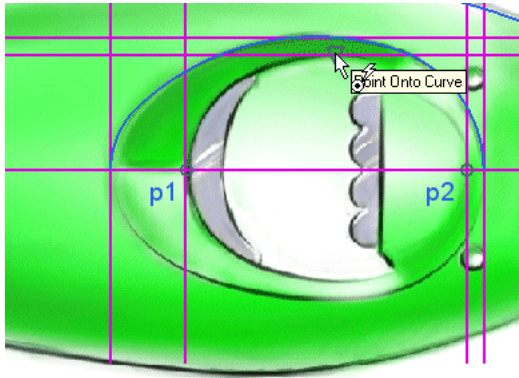
- Start the **Three-Point Circle** command. Change the Snap Point option to **Tangent Snap**.
- Choose the first point on left end circle.
- Select the second point on right end circle with **Tangent Snap** option.
- Select the third point in such a way that, the curve matches the conical shape on the image.



Use **Trim/Extend Curves** and **Smart Delete** command to delete redundant entities.

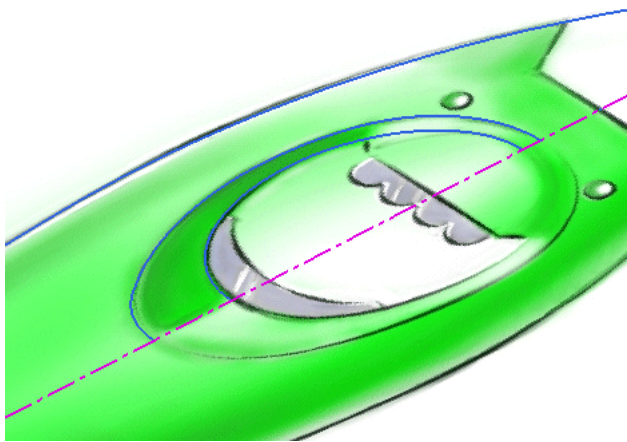
Let's draw the inner conic now.

- Go to the **Conic Curve** command.
- Set the Type option to Ellipse: 2 vertices and 1 point
- For the first two points, select the **Intersection Point Snap** of the axis and the two vertical lines that define the inner conic shape.
- For the third point, change the Snap Point option to **Point on Curve Snap**. Click a point on the horizontal line drawn at the uppermost point of inner conic.



Smart Delete the lower half of this inner conic curve. Hide all the construction curves shown in magenta color except the axis line.

Change the **Line Type** to #4 (Centerline) for the axis line.



This completes the basic curves needed for the top view of the model. Let's now draw the basic curves for the front view.

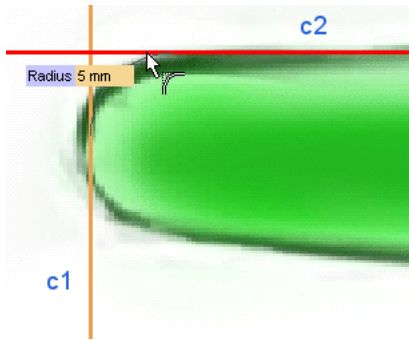
We consider the upper view to be the top view and the lower view to be the front view. Lets start laying out the outline of the front view.

- Go to **Insert** → **Drafting** → **Line** → **Angular** command. Set the angle to Vertical 0 deg to want the support as flat.

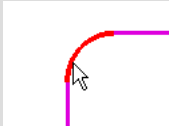


Zoom-out until you see the vertical left-limit line. Extend this line until it overlaps the front view. See image below for reference.

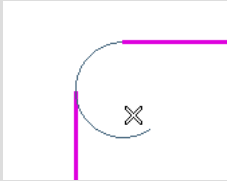
- Go to the **Insert** → **Drafting** → **Fillet** command.
- Set Trim/Extend Curves as None.
- Choose the Base line from front view and the vertical extended line to insert a fillet.
- Judge the corner as seen in the image below and enter the final Radius value.



Note: How extend the arc's length.

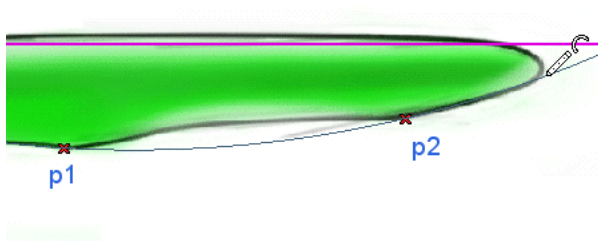


Double click on the entity on the side you want extended.



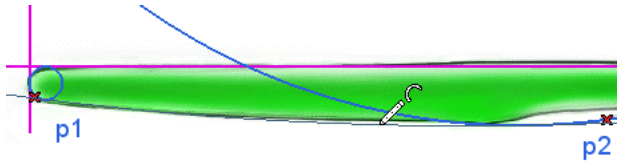
Move the cursor until end condition and pick the graphic area.

- Zoom in to the right side of the front view so that we can move ahead in tracing front .
- Start the **Three-Point Circle** command. Set the mode to Circle. Insert the first point on the lowermost point of the front view.
- Select the other two points such that the new curve matches the outer boundaries of the image as shown below.

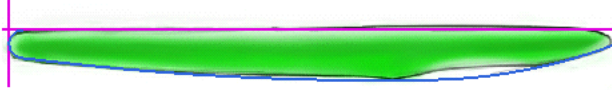


Now draw the third circle to bridge the gap between the last two circles we have just drawn.

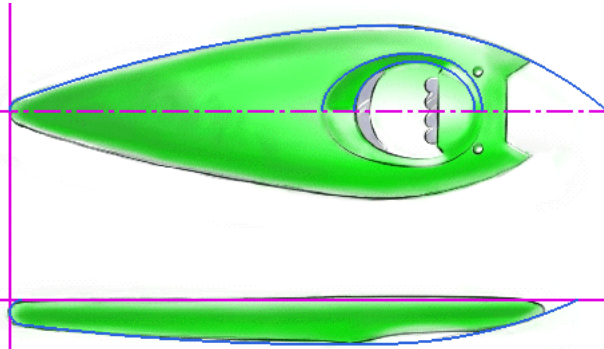
- Go to **Three-Point Circle** command. Set the mode to Circle. Insert first point on the lowermost point of the front view.
- Set the Snap point option to **Tangent Snap**.
- Select the initial two points using Tangent Snap option so that the new curve will be touching these two circles tangentially.



Use the **Smart Delete** and the **Trim/Extend Curves** commands to delete all the redundant curves and keep only those which matches the boundary according to the image below.



So, the final views should look like this.



In this first Step we have defined the main curves that represent our object. You'll find same in a other tutorial but moved in 3d space.

2. Step 2 - More Curve Creation on a new model

In this step we will start a new concept sketch: one of a biscuit container. We will examine it's shape and determine the best method for curve creation. First, let's set up our modeling environment for this task. Go to **New Model** to create a new model file, and set the default units to millimeters (if not already).

Next, we will insert a rectangle that represents the final size of our product.

- From the drafting toolbar, select the **Rectangle** button. The Rectangle selection list appears in the modeling window.
- In the Selection list, change the Mode to Mode: Cen+Size
- Set the size to X size430 mm, and Y size335 mm
- Snap the rectangle to the **Work Plane Origin** to center it on the WP origin.



- Click on the **Insert Image** button to bring in a concept sketch of our Biscuit Holder.
- The image file resides in the files folder inside the task folder of the downloaded webtask.
- Select the Biscuit Holder.jpg image and open it.

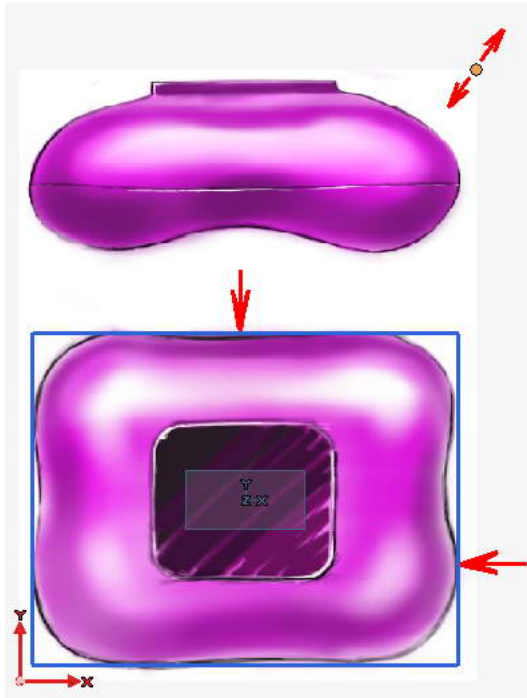
thinkdesign prompts you to select an insertion point for the image.

- Pick the lower left corner of the rectangle.



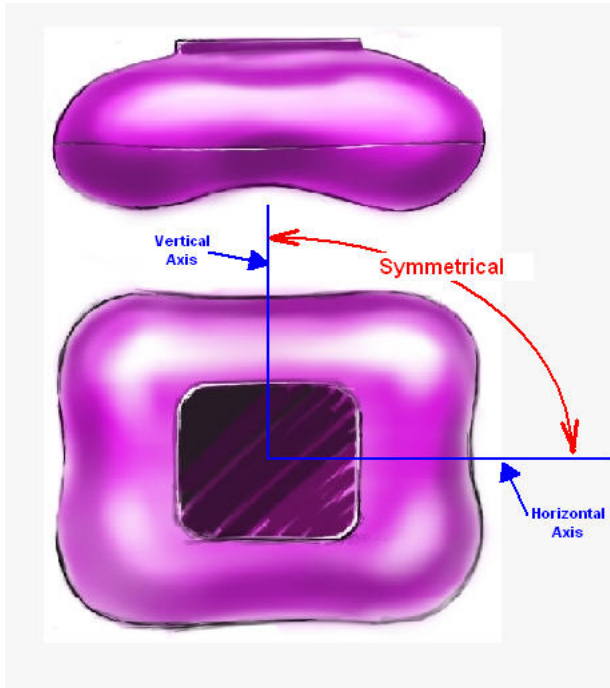
Now that we have our image in our modeling window, let's move it and resize it so that the top view of the biscuit container fits into our rectangle.

- Click on the **Modify Image** command from the Image toolbar.
- Use the Red Dot or "Handle" at the center of the X and Y axis to move the image around. You can use the X and Y axes as handles to rotate the the image.
- Use the Red Dot or "Handle" in the upper-right corner of the image to scale the image up and down.
- Fit the image into the blue rectangle as shown below.



OK, That's great! Now let's evaluate the shape of the product that we need to model.

- Using **Hide Entities**, hide the Blue Rectangle so that we can get a better look at the concept sketch.



OK, that's good. From the Top View, the biscuit holder is rectangular and symmetrical about the vertical and horizontal centerlines or axes. So from a Top View, we could actually model only one quarter of the object and then mirror the other three corners of the model. This is a great time saving technique and we'll use it in this task.

The side view also looks symmetrical from the left side to the right side. Modeling one quarter of this part and mirroring the remaining areas should definitely work well here also.

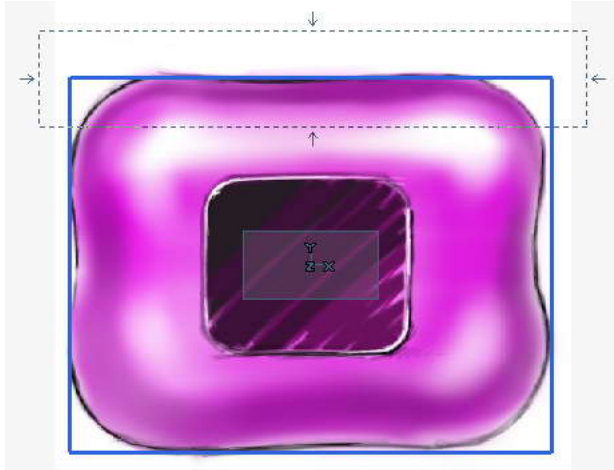
3. Step 3: Construction Curves

In this step we will continue to create construction curves and lines. These construction lines will help us build the framework for our surfaces. These steps are critical to ensure that our end products fit our overall size and it's shape is true to the concept sketch. Let's get started!

- Select **Unhide Entities** and unhide the blue construction rectangle so that it's visible now.

Next, we will select the three upper lines on the triangle. Then we will Drag and Copy these three lines to create a second rectangle on top of our original one. Let's see how this is done.

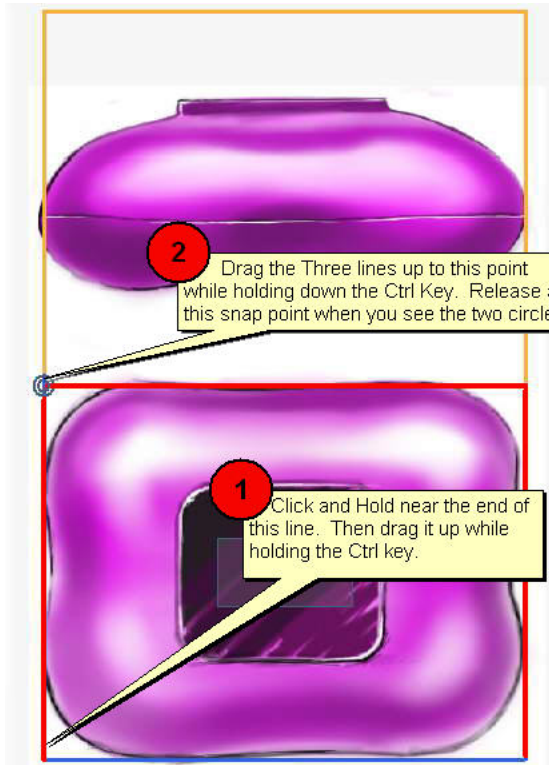
- Click and drag a selection window from right to left in the area shown.



- With your Left Mouse Button (LMB), Click and Hold NEAR the bottom end of the left vertical line.

You'll see a circle at the end point indicating that the end will snap to another point when you release the mouse.

- Drag these three lines up to the top left point of the rectangle while holding down the **Ctrl** Key. Holding the Ctrl Key while dragging creates a copy.
- Release the LMB at this top left snap point when you see the two circles.

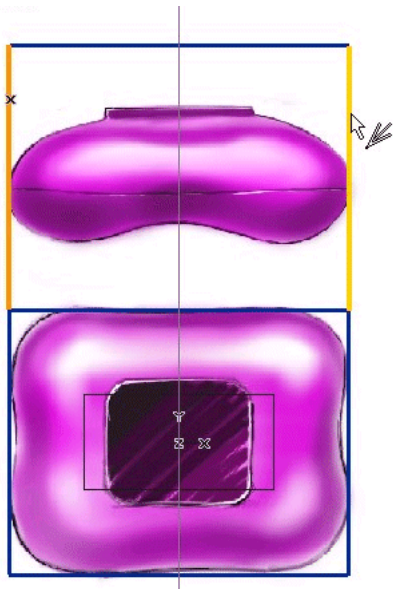


OK, next we'll be adding construction lines to both the views.

- From the drafting toolbar, select the drop-down arrow next to the 2 point line command. Select **Bisector** from the drop down menu.

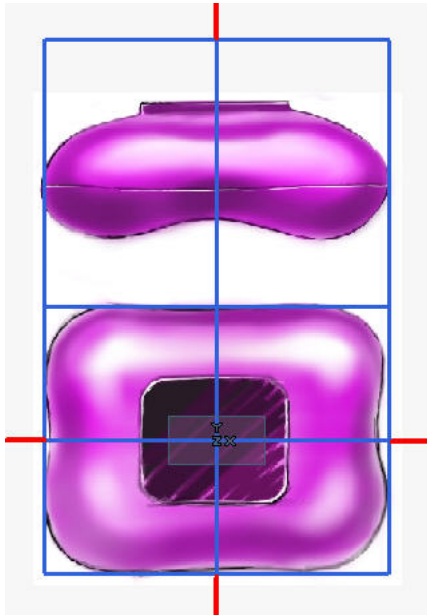
In the Bisector line selection list, change the type to Unlimited.

Select the two vertical lines on either side of the sketch. A preview is generated.



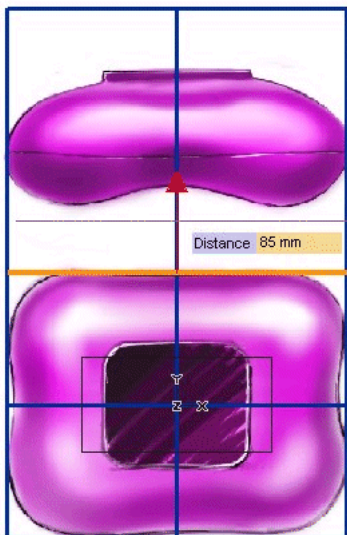
- Select the two lower horizontal lines to create a horizontal centerline.
- Hit **Esc** or Cancel to exit the command.

- Start the **Smart Delete** command and trim off the excess lines as shown below. (trim the red pieces)
- Hit **Esc** or **Select Entity** to exit the command.

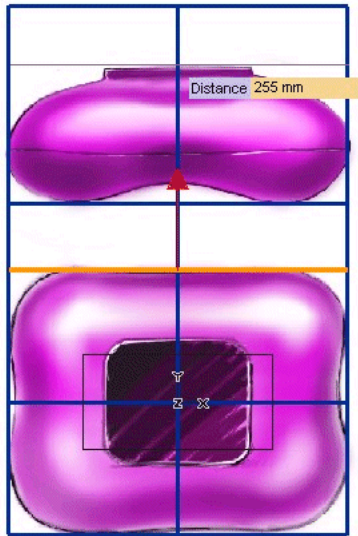


Next, we'll focus on creating construction lines for the end view.

- Start the Parallel line command. Select the middle horizontal line and move it up by 85mm so that the parallel line is at the bottom of the side view sketch. Double click on the red arrow if it is facing the wrong direction.

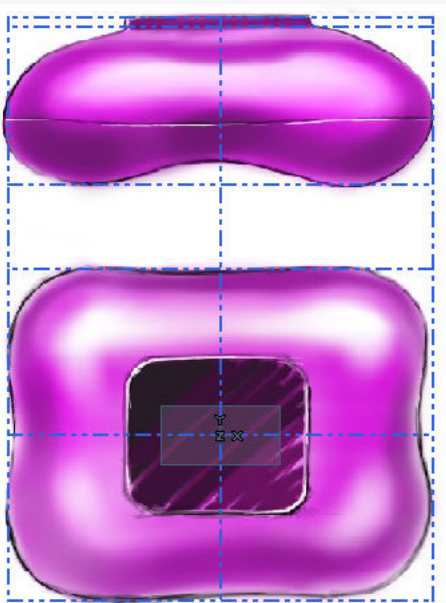


- With the Parallel line command still active, create another line, 255mm above the center horizontal line. This line should lie at the top of the side view sketch. See the image below for reference.



- Again, with the Parallel line command still active, create another line, 245mm above the center horizontal line. This line should lie at the top of the side view sketch.
- Hit **[Esc]** or Cancel to exit the command.
- **Select All** the lines and right click in the graphics area to pull-up the context menu and select **Properties**.
- Change Line Type to #5.
- Change the **Line Width** to line #2.

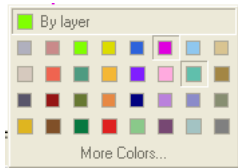
The modeling window should look similar to the image below.



Now let's create more construction lines...but we need to change our default color so that they are much easier to identify.

- Hit **[Esc]** or **Select Entity** to make sure nothing is selected.

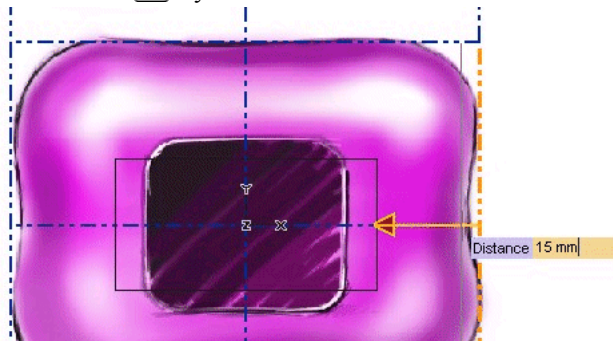
- Go to **Options/Properties** and under the Entity Properties tab change the color to #3 -- which is a green color.



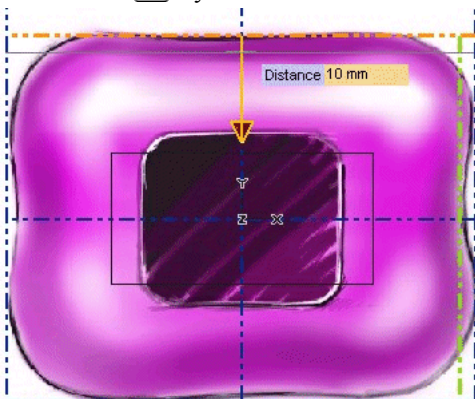
- Change the line type to #5.

Now we will focus on the upper right corner of our plan view. First, we need to create some parallel lines to define the indentations on the sides and top of the part.

- Start the **Parallel Line** command. Make sure the Type is set to Unlimited.
- Select the right vertical line.
- Move your mouse until the Mini-dialog shows Distance and in the minidialog insert a value of 15 mm. This will insert the line -- but make sure it is on the inside. You can also invert the direction of parallel line towards inside at any time by double clicking on the red arrow.
- Press the **Esc** key to exit the command.

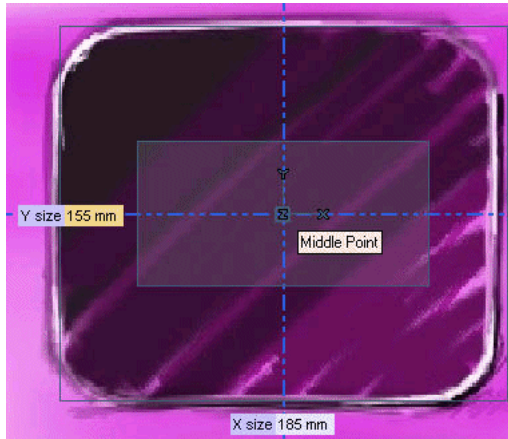


- Press **Enter** to repeat the last command i.e., **Parallel Line**
- Select the horizontal line at top of the plan view.
- Move the cursor below the selected line by Distance 10 mm.
- Press the **Esc** key to exit the command.



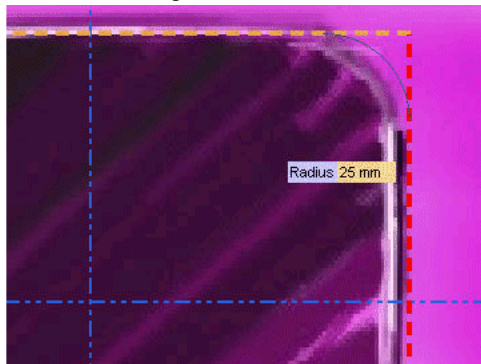
Now we will work around neck area of our object.

- Start the **Rectangle** command.
- Press the **[z]** key and drag a selection box around the top view to zoom in.
- Create a 185mm X 155mm rectangle centered on the horizontal and vertical Centerlines as shown below.



This rectangle will help us to identify the neck area of our Biscuit Container. Let's get the corners rounded.

- Start the fillet command and select the right vertical line of the rectangle we just created.
- In the Mini-dialog, change the radius size to 25mm.
- Select the top horizontal line of our new rectangle to fillet the two lines.

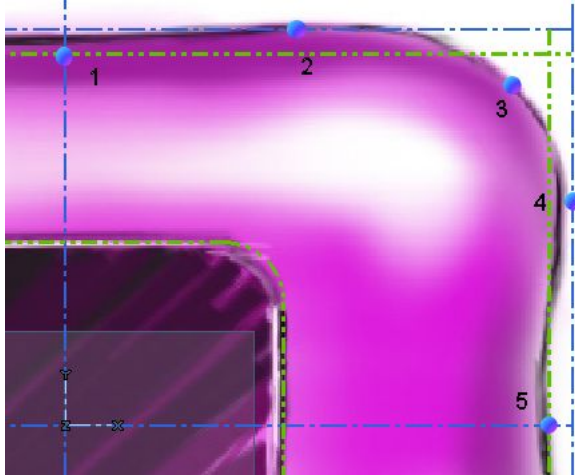


Now is time to create the real geometry to give a shape to our product. We will use the **Insert Curve through Interpolation Points** command, an appropriate tool to make geometry given the exact points of passage. You have to select the marked point in the below image.

Before you begin set the Color to #5, Line Type to #1 and Line Width to #3.

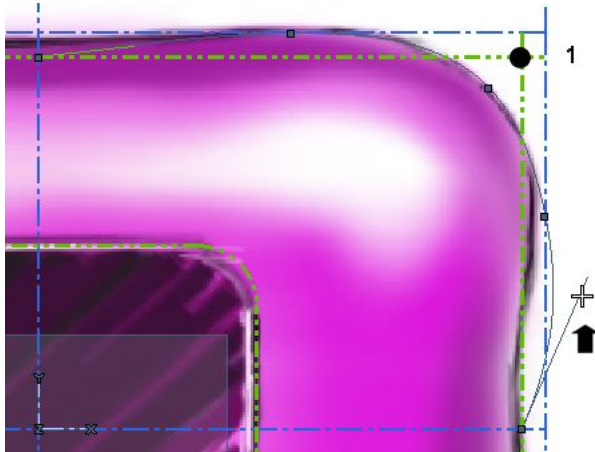
- Start **Insert Curve through Interpolation Points**
- As first point use **Intersection Point Snap**.
- As second point use **Point on Curve Snap** on the horizontal limit.
- As third point use a free point over the image.

- As fourth point use **Point on Curve Snap** on the vertical limit.
- As fifth use **Intersection Point Snap**.

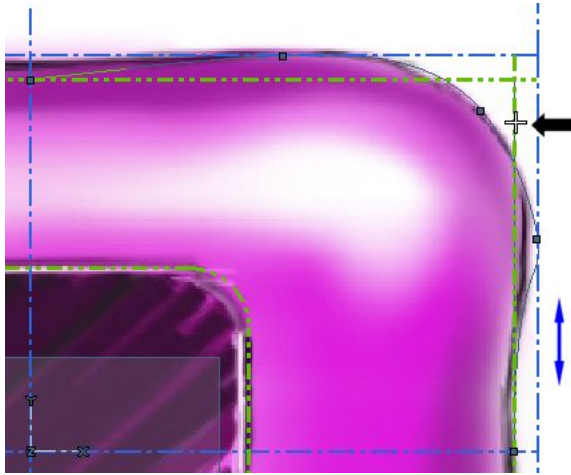


thinkdesign shows a preview as first result. Now you have to change the start and end tangent vectors to assign the better condition with next mirror parts.

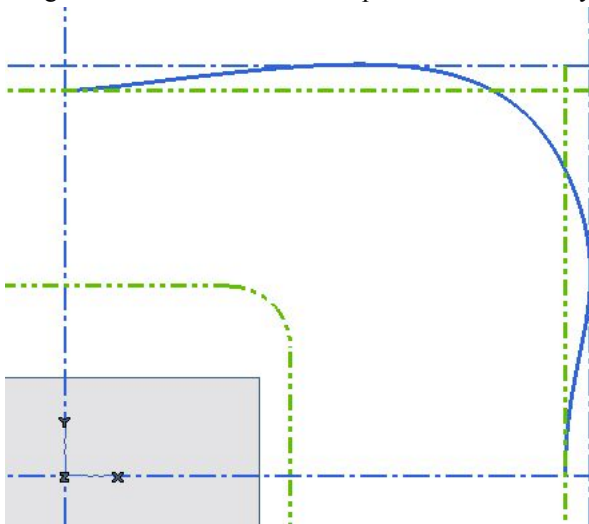
- Select with the right mouse button the end tangent vector and choose Set Tangent.
- Move the cursor to intersection point 1. The vector follows the mouse cursor.



- Select again, with right mouse button, the guide vector, now covered by green vertical line, and choose Set Tan Magnitude.
- Move the cursor up and down until you get the required shape.

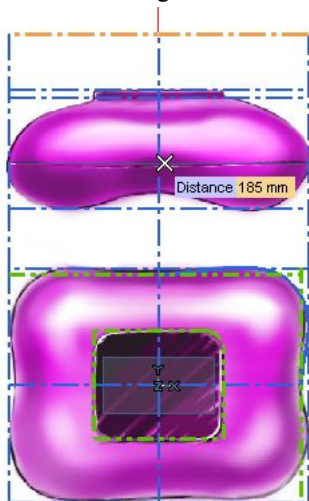


If you want to change the new curve, start the command **Modify Curve Interpolation Points** and select it. The image shown below is one of the possible results that you might get.

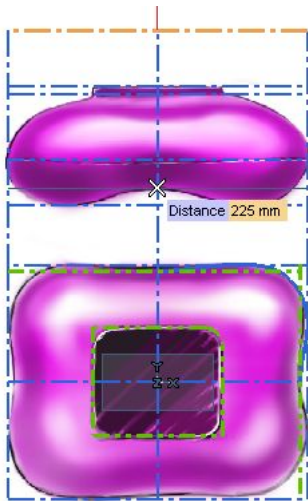


Let's work on the end view to get our initial curves.

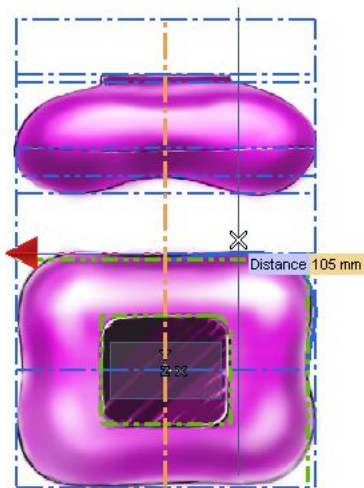
- With the Parallel line command create one more line below the top horizontal line, at a distance of 185mm. See the image below for reference.



- With the Parallel line command active draw another line at a distance of 225mm.



- Now take the middle vertical axis as reference and draw a parallel line at a distance of 105mm.

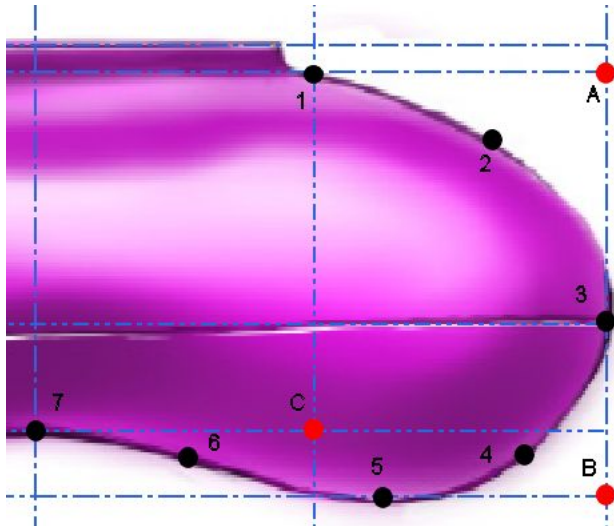


You can change the Color, Line Type and Width of the curves at any time. Pre-select the curves (hold the shift button for more than one entity selection) and right click in the graphics area. In the context menu, select **Properties** and make changes in the Entity Properties dialog window

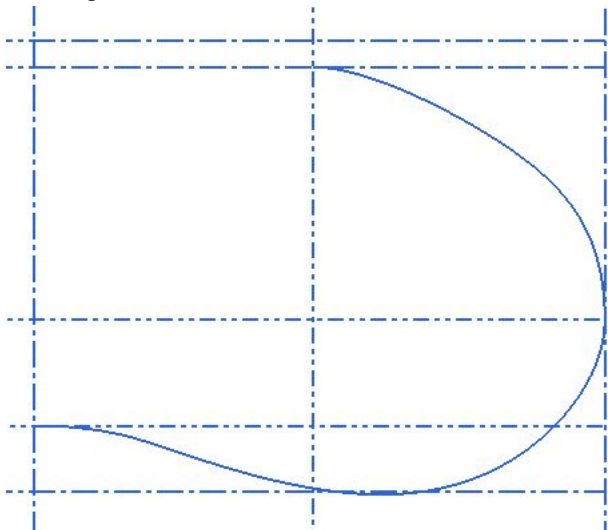
Good!! You have created the reference curves! Zoom in to the side image and make the other two curves using the **Insert Curve through Interpolation Points** command.

- Start **Insert Curve through Interpolation Points** to make the first curve using 1 - 2 - 3 marked points as shown in the image below.
- As first point use **Intersection Point Snap**.
- As second point use a free point over the image.
- As third use **Intersection Point Snap**.

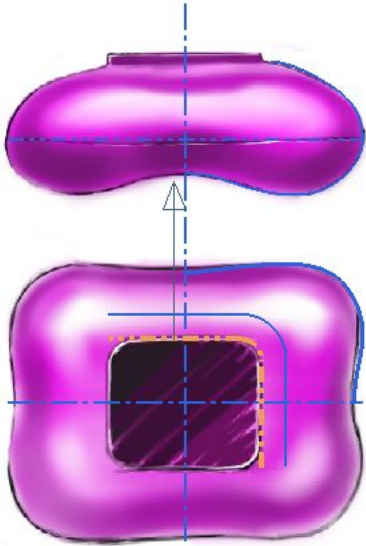
Use marked point A to impose tangent direction for both vectors.



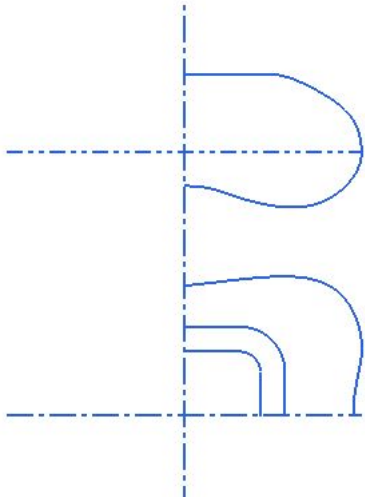
- Again start the **Insert Curve through Interpolation Points** for the second curve with 3 - 4 - 5 - 6 - 7 points as shown in above image.
- As first point (3) use **Intersection Point Snap**. You'll use point B for tangent vector.
- As second point (4) use a free point over the image.
- As third point (5) use **Point on Curve Snap** on the bottom horizontal limit.
- As fourth point (6) use a free point over the image.
- As fifth (7) use **Intersection Point Snap**. You'll use point C for tangent vector. You can see the result in the image below.



- Start **Insert** \rightarrow **Drafting** \rightarrow **Offset on Plane** command.
- Select the three curves that represent the neck.
- Impose 30 mm as distance. See the second image below for reference.



- Use **Hide Entities** and **Smart Delete** to have the entities as base for next step.



Clean up the model to have only the above shown entities. Now we are ready to start creating the model!!!