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## **Introduction**

### **CATIA Version 5 Electrical Space Reservation**

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

- Defining reservation areas for electrical equipment
- Defining reservation routes for electrical pathways
- Importing reservation areas
- Analyzing the space reservations for clashes and clearance issues

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## Manual Format

It is important to understand the format of the manual in order to use it most effectively. This manual is designed to be used along with an instructor; however, you will need to do a lot of reading as well, in order to fully understand CATIA Version 5. The exercises in this book will list steps for you to complete, along with explanations that try to inform you what you have just done and what you are getting ready to do. The actual steps are in bold type and the information that follows the steps is for your benefit. Anything that appears in *italics* refers to a message CATIA provides—this includes information in pull-down menus, pop-up windows and other messages.

An example of a step and its explanation is shown below (note: normally the lines will not be there):

---

**Select a location to the right of the origin.** This specifies the other end point of the line. You will continue specifying locations in order to complete your profile. It should appear similar to the diagram shown below.

---

As you can see, the desired action blends in with the text except that it appears in bold. The information following the step explains what that step accomplished and where you are going next. It is important for you to read this information to help in your understanding of CATIA Version 5.

Also, you will find that the exercises build upon themselves. Later exercises often assume you know how to do certain steps which have been covered in earlier exercises. If you did not quite pick up what you needed to know from an exercise, you will probably wish to review it several times before moving on to the more advanced sections. As you progress through the manual, it expects that you are learning and therefore you are able to do a lot more with fewer steps. Eventually, you are expected to be able to perform previous tasks without any steps.

## **Electrical Reservations**

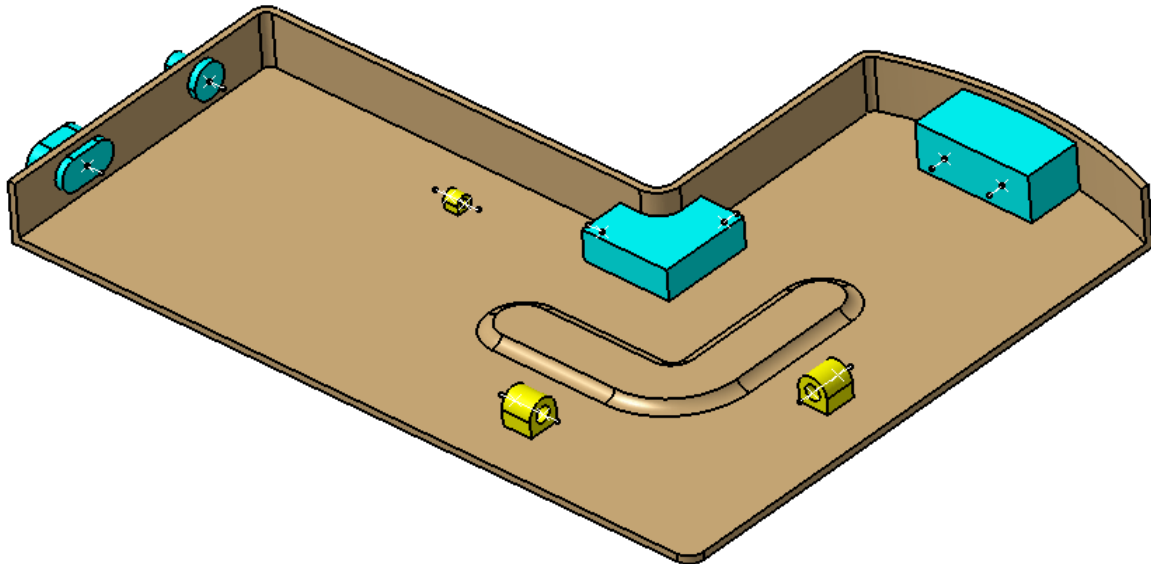
Creating an electrical reservation involves creating shapes and areas that will act as a keep-out zone or an area to avoid for other disciplines. Although CATIA V5 has a product for Systems Space Reservations, you are limited to creating reservations that are only simple shapes. Many times you will have a complex shape to conform to and will be unable to accurately model and describe the space reservation through blocks and circular tubes. It is for this reason that most of the time space reservations are made via part design and wireframe shapes and objects. These objects allow for a more complex definition of the area to be reserved.

This course builds upon your knowledge of Part Design and Sketcher, Wireframe and Surfaces, and Assembly Design. All three of these topics will be used to define the electrical reservations.

## Pathway Reservations

Pathway reservations are necessary for ensuring that enough space is left over for any paths between equipment that will be installed at a later date. Pathway reservations can be used for any type of part. They are generally used for electrical wire routing or tubing paths.

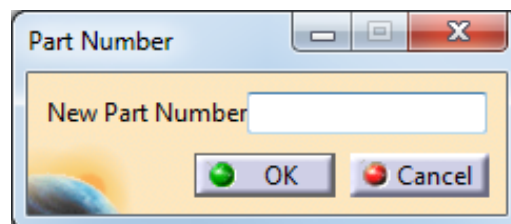
Open the **Pathway Reservation** document from the *Pathway Reservation* directory. The model appears as shown.



This model has already been started for you. This model was done exactly like the Equipment Reservation exercise that you just completed. You will build pathway reservations between the equipment and support elements.

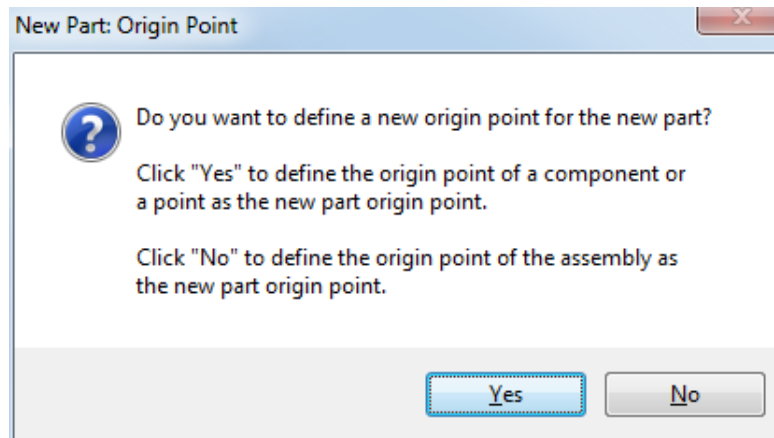
**Be sure that you are in the Assembly Design workbench.**

Select the **New Part** icon and select *Pathway Reservation* from the tree.  The *Part Number* window appears.

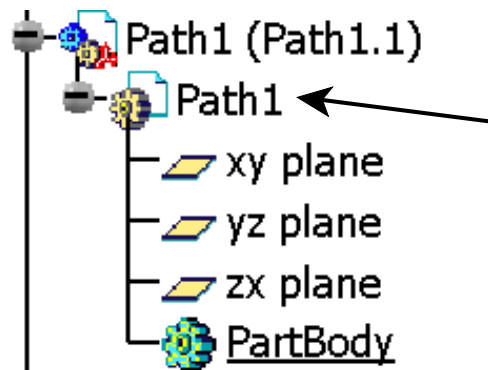


Key in **Path1** for the *New Part Number* and select **OK**.

The *New Part: Origin Point* window appears.



Select **No**. You will just use the global origin point of the assembly. The new part appears in the specification tree.



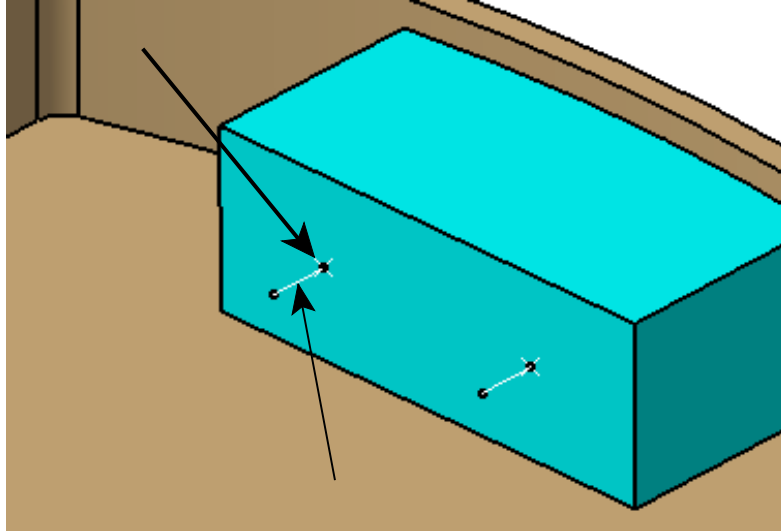
**Double select the *Path1* document as shown above to activate the part.** This should switch you to a part level workbench. You are going to build this part within the context of the assembly.

**Switch to the Generative Shape Design workbench.** You will be using wireframe to create the paths.

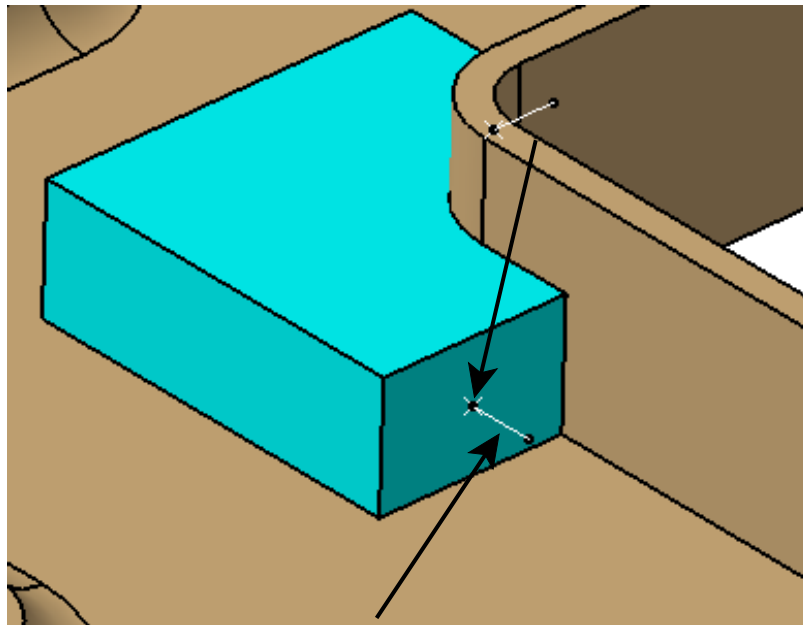
**Hide the reference planes of the part.** You will not need the reference planes.

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

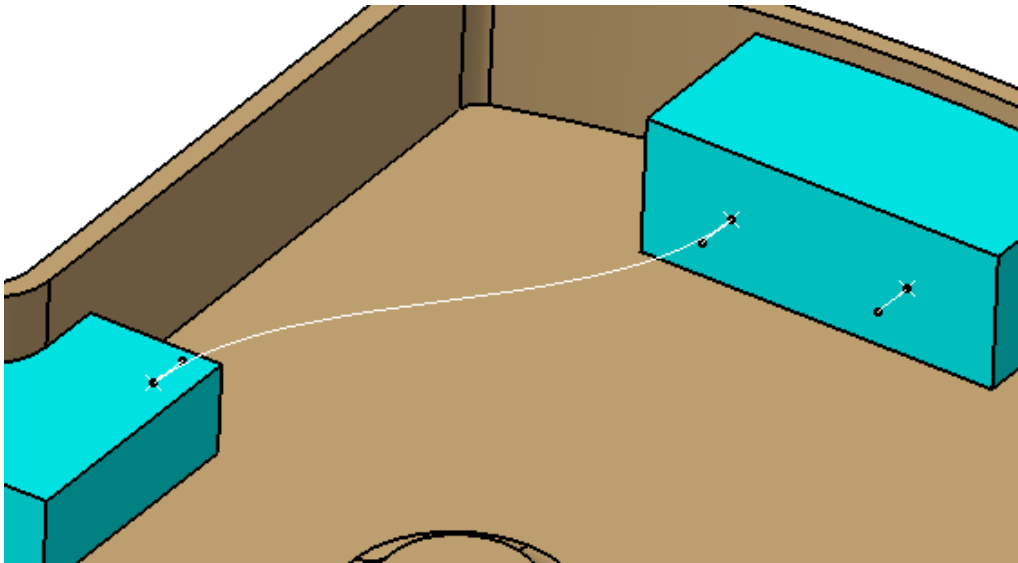
Select the point and line as shown below to define the start location and tangent direction of the spline. Be sure that the tangent direction arrow is pointing out of the box.



Select the point and line as shown below to define the end point and tangent direction of the spline. Be sure that the tangent direction arrow is pointing into the solid.



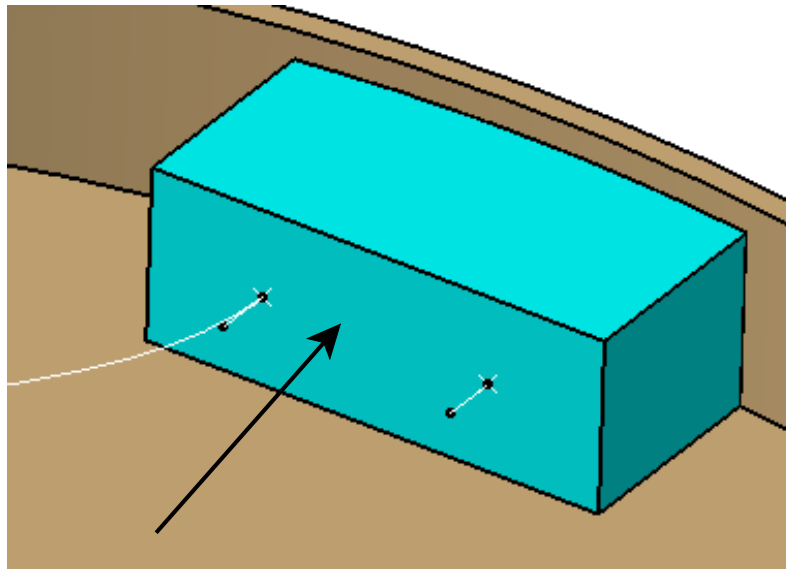
Select **OK**. The spline should appear as shown.



You will create a few different pathways so we will look at some different options for doing each path.

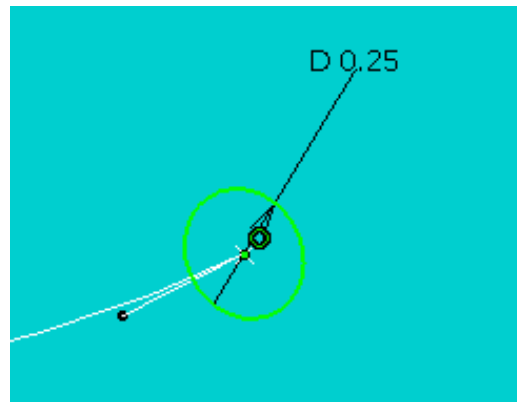
**Switch to the Part Design workbench.**

**Create a sketch on the face as shown below.**






**Draw and constrain a circle as shown below.** The center point of the circle should be constrained to the point.

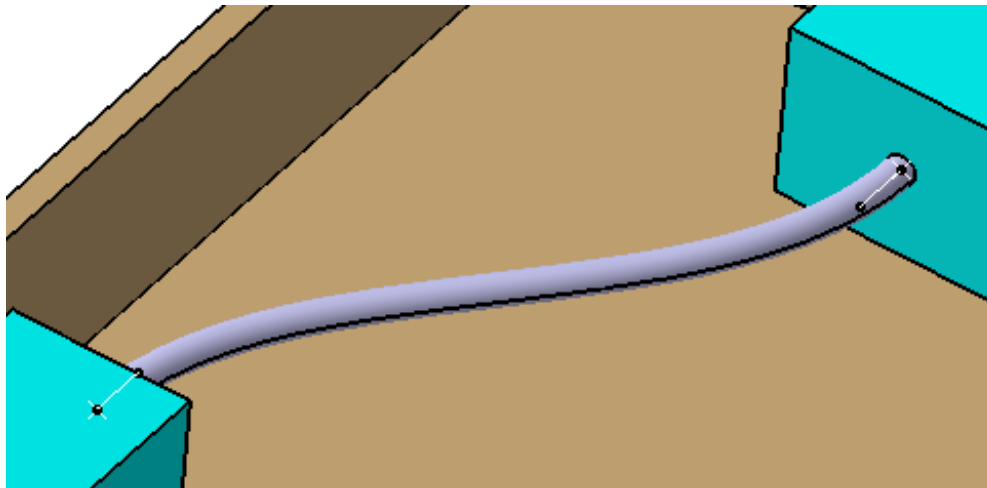


**Exit the sketch.**

**Select the Rib icon.**  You may get a warning telling you that you need to be in a partbody to create a solid feature. Just select *OK* to the *Warning* window.

**Select the sketch of the circle to define the *Profile* and select the spline to define the *Center curve*.**

**Select *OK*.** The rib is created as shown.



This will represent a pathway reservation between the two equipment reservations.

**Double select on *Pathway Reservation* in the specification tree to activate the product level.**

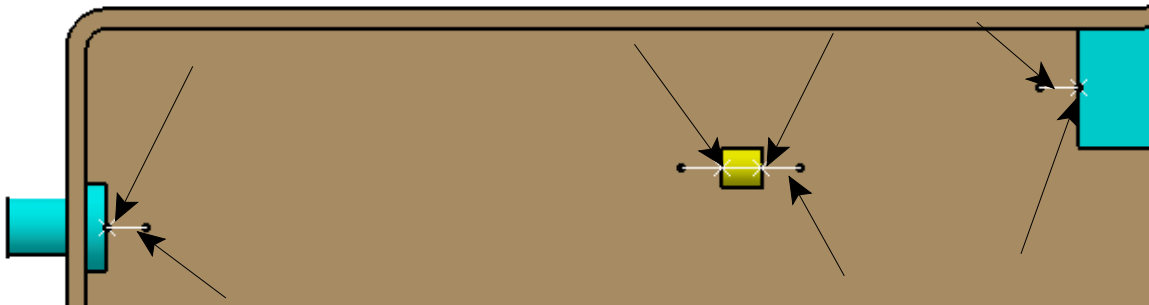
**Use *Save Management* to save the assembly and all of the parts in a folder called Pathway Reservation Assembly in your directory.**

Create a new part called **Path2** that uses the global assembly axis as its local axis.

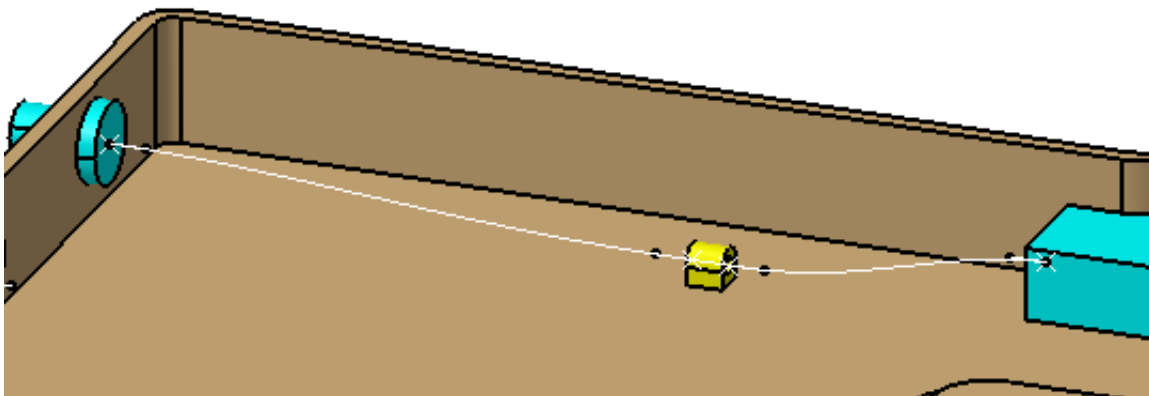
Activate the *Path2* document and hide the reference planes of the part.

Switch to the Generative Shape Design workbench.

Create a spline using the points and directions shown below.



It should appear as shown.



This time you will create a surface to create the solid pathway.

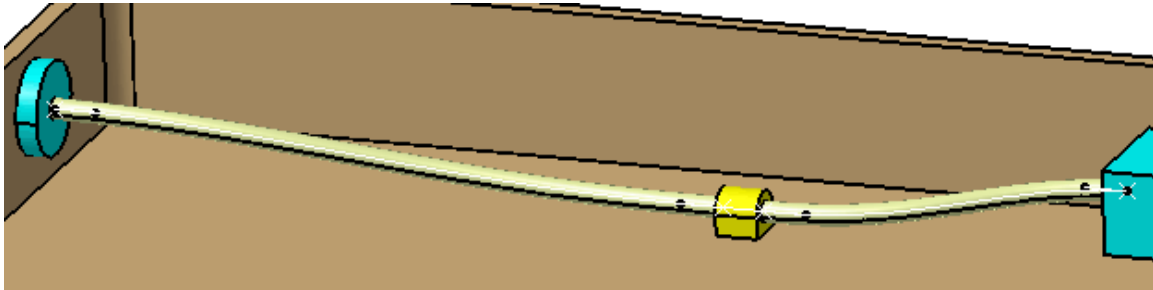
Select the **Sweep** icon.  The *Swept Surface Definition* window appears.

Select the **Circle** icon in the window to define the *Profile type*. 


Change the *Subtype* to *Center and radius*.

Select the spline to define the *Center curve* and set the *Radius* to be 0.125.

Select **OK**. The surface should appear as shown.

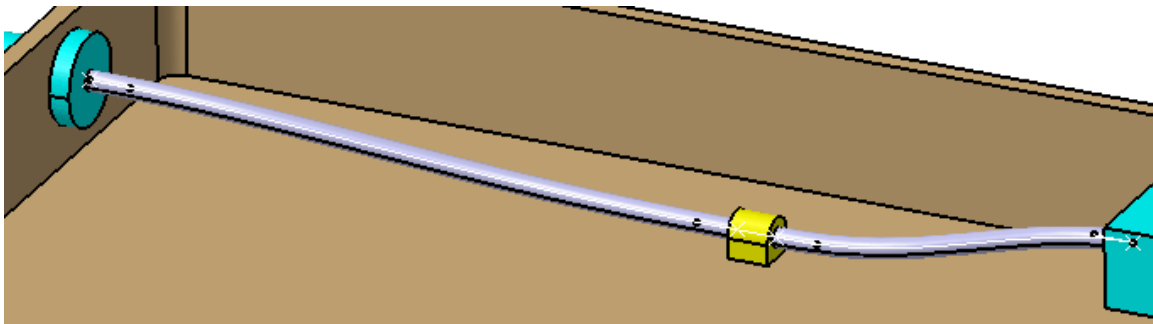


Switch to the **Part Design** workbench.

Select the **Close Surface** icon.  The icon is located under the Thick Surface icon. Again, you may get a warning. Just ignore it. The *Close Surface Definition* window appears.

Select the sweep that you just created and select **OK**. The solid is created.

Hide the sweep. The model should appear as shown.

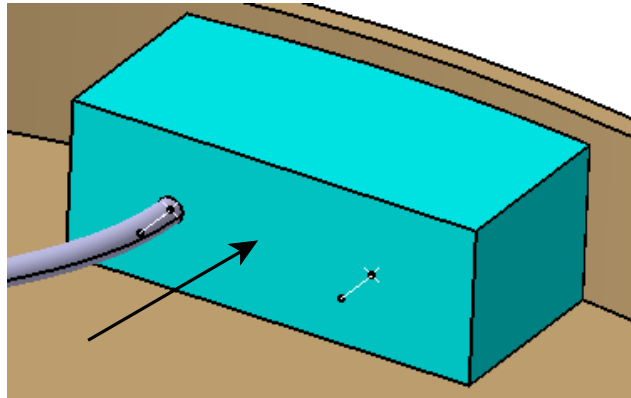


Double select on *Pathway Reservation* in the specification tree to return to the product level.

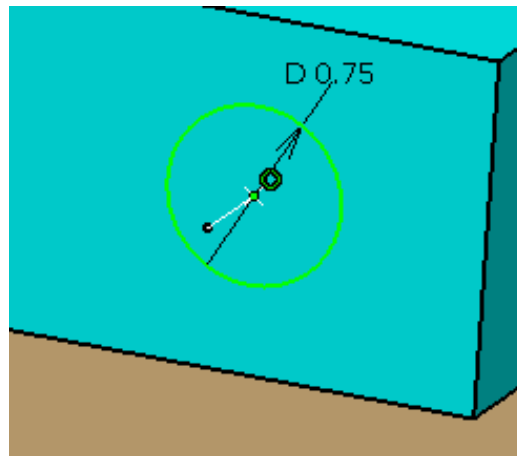
Create a new part called Path3 in the same manner as before.

Activate the part so that you can build within it and go ahead and hide the reference planes as well.

Create a sketch on the face shown below.

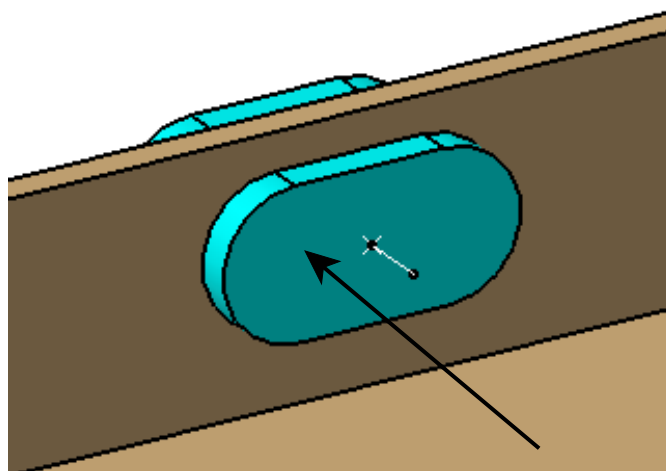


Draw and constrain a circle as shown below. The center point of the circle should be constrained to the point.



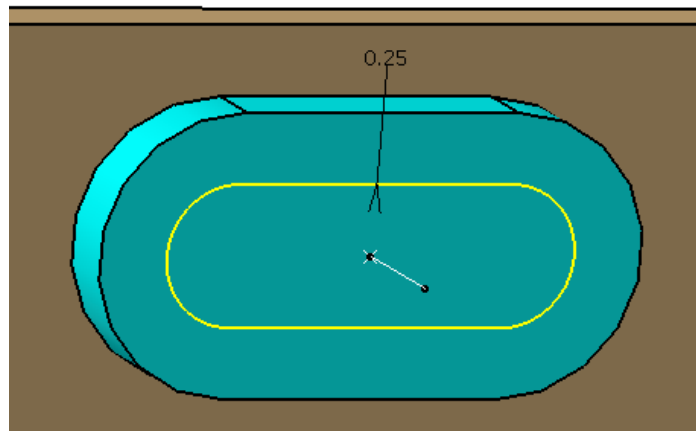
Exit the sketch.

Create another sketch on the face shown below.



Select the **Offset** icon and select the same face again. 

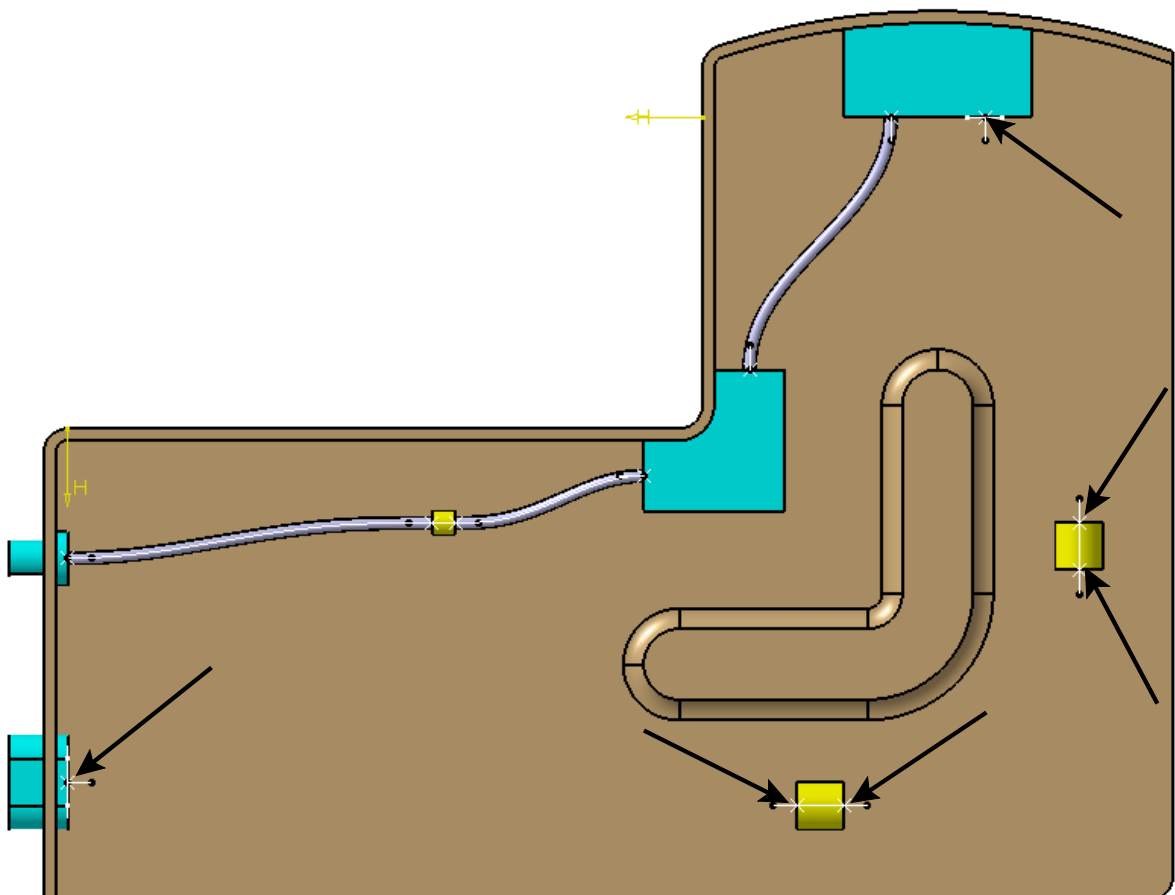
Set the offset to be **0.25 inches**. The sketch should appear as shown.



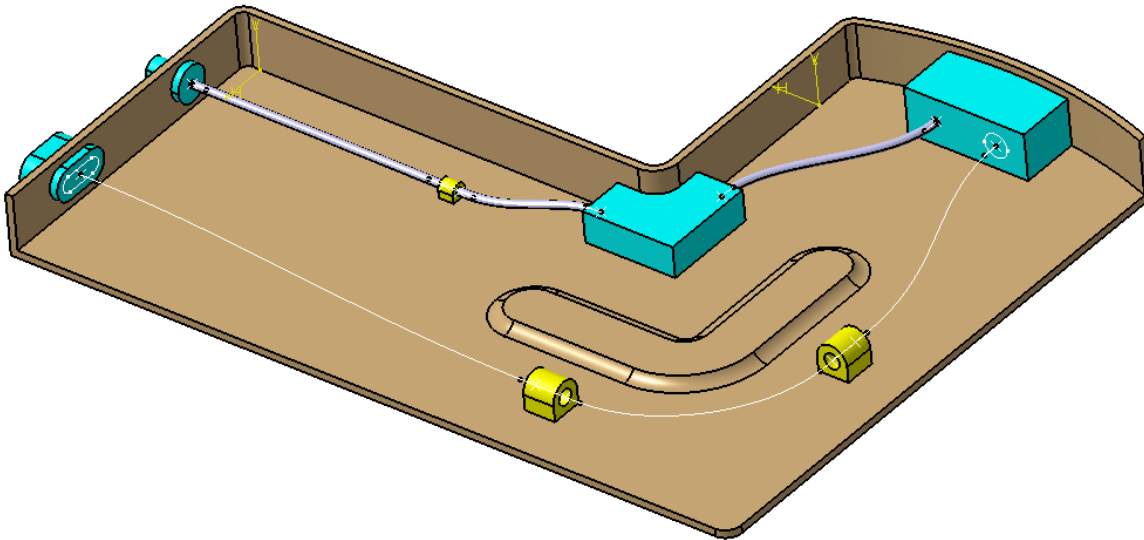
Exit the sketch.

Switch to the **Generative Shape Design** workbench.

Create a **spline** passing through the points as shown and using their respective directions.



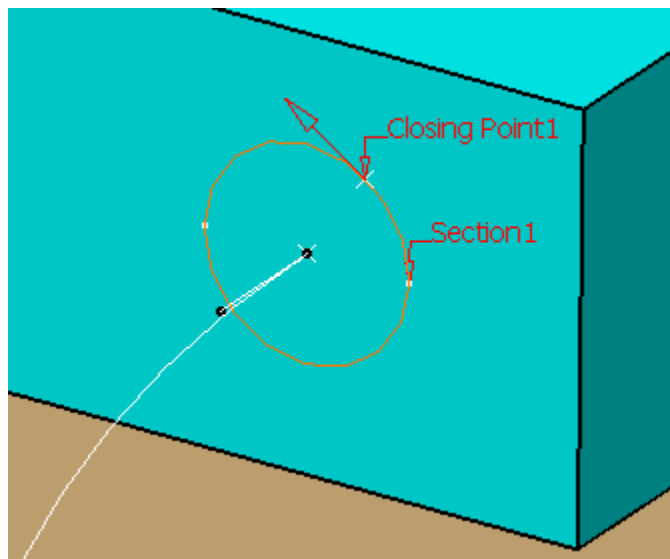
The spline should appear as shown.



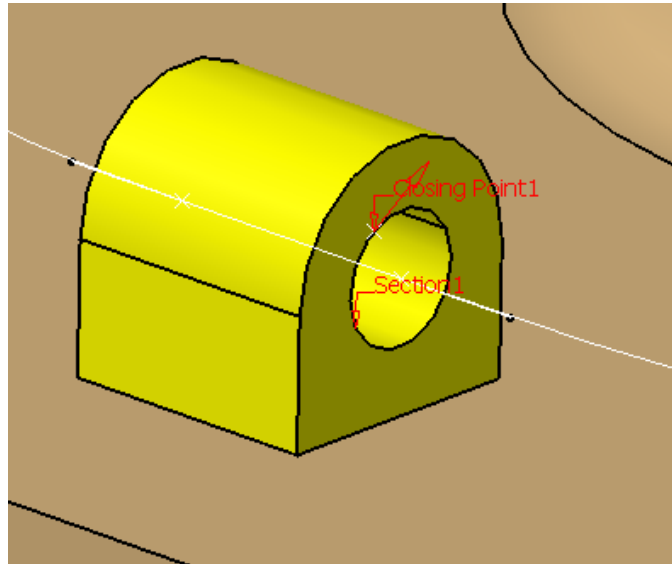
Switch back to the Part Design workbench.

Select the **Multi-Sections Solid** icon.  The *Multi-sections Solid Definition* window appears.

Select the circle to define the first profile. Make sure the *Closing Point* arrow is pointing in the direction shown below.

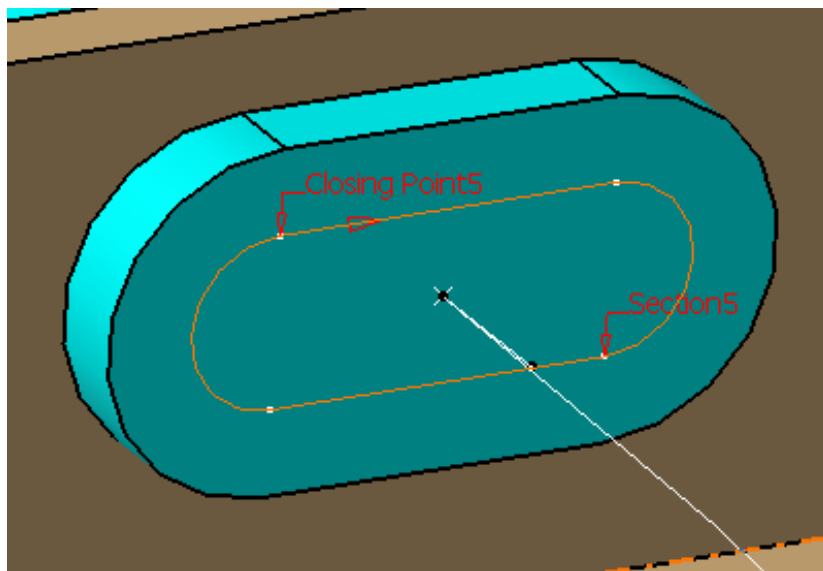


Select the circular edge as shown to define the second profile. Make sure the *Closing Point* arrow matches the picture.



Select each of the next three circular edges to define the profiles on the two yellow supports that the spline passes through. Make sure the *Closing Point* arrows are all pointing the same direction.

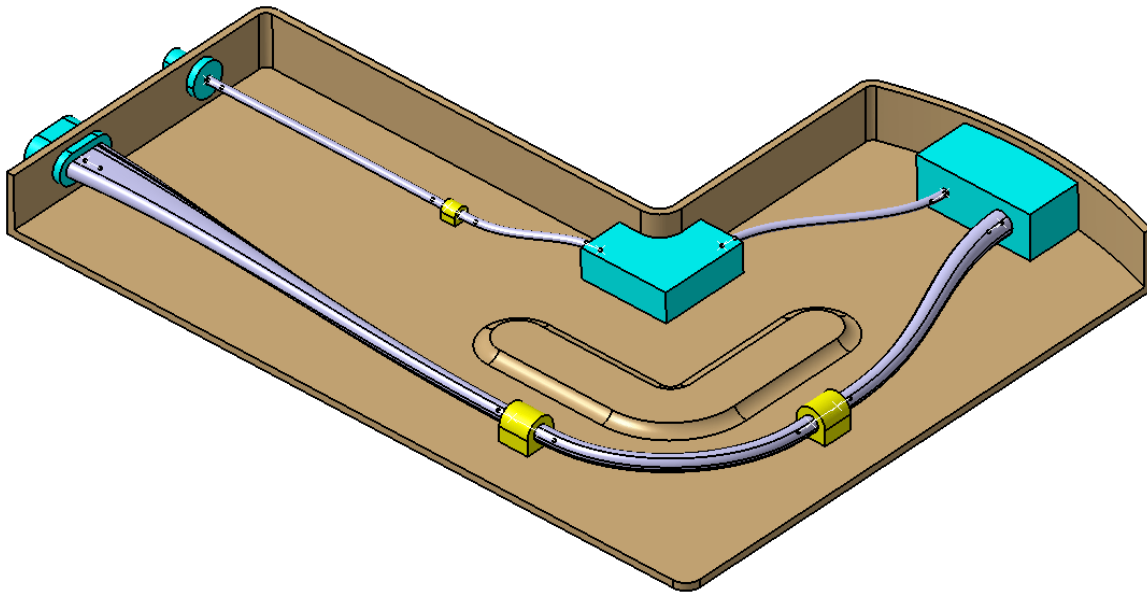
Select the elongated hole sketch to define the last profile for the multi-sections solid. Again be sure the arrow is pointing as shown.



Select the *Spine* tab in the window.

Select the spline in the display to define the *Spine* for the multi-sections solid.

Select the *Coupling* tab and change the *Sections coupling* to be *Ratio*. Select *OK*. The path should appear as shown.



Double select on *Pathway Reservation* in the specification tree to activate the assembly. You may want to save your assembly at this point as well.

Create another part in the assembly called Path4.

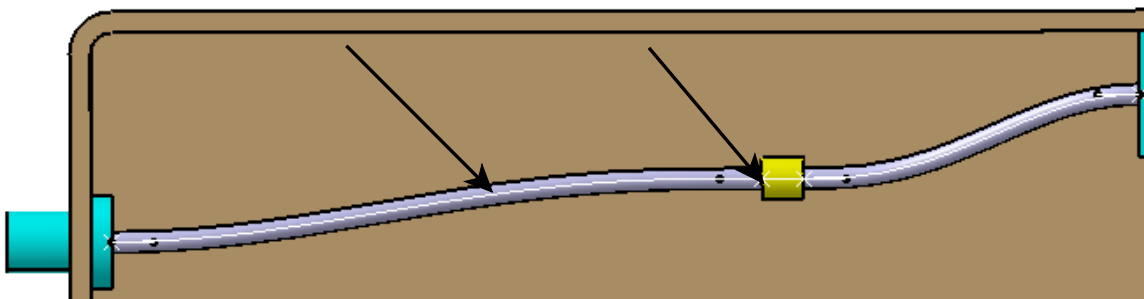
Activate the part and hide the reference planes.

Switch to the *Generative Shape Design* workbench.

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

Change the *Point type* to *On curve*.

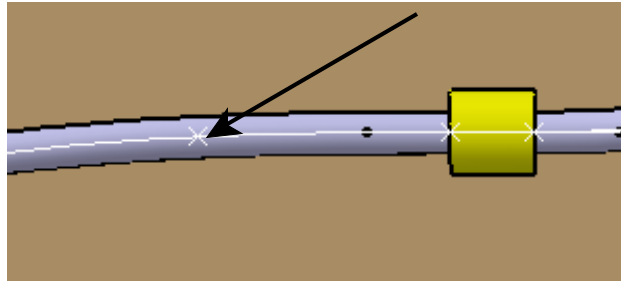
Select the spline shown below to define the *Curve* for the point operation.



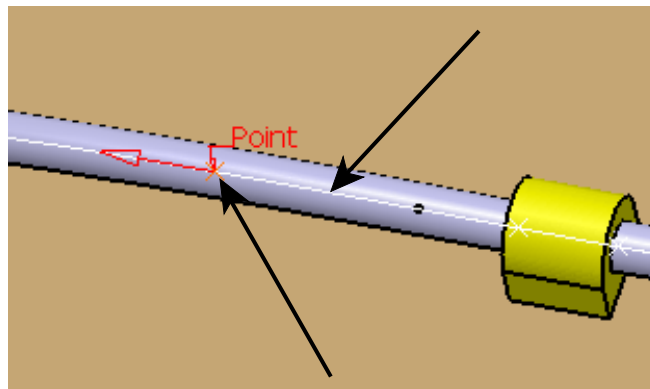
Select in the *Reference Point* box in the window and select the point as shown above to define the *Reference Point*.



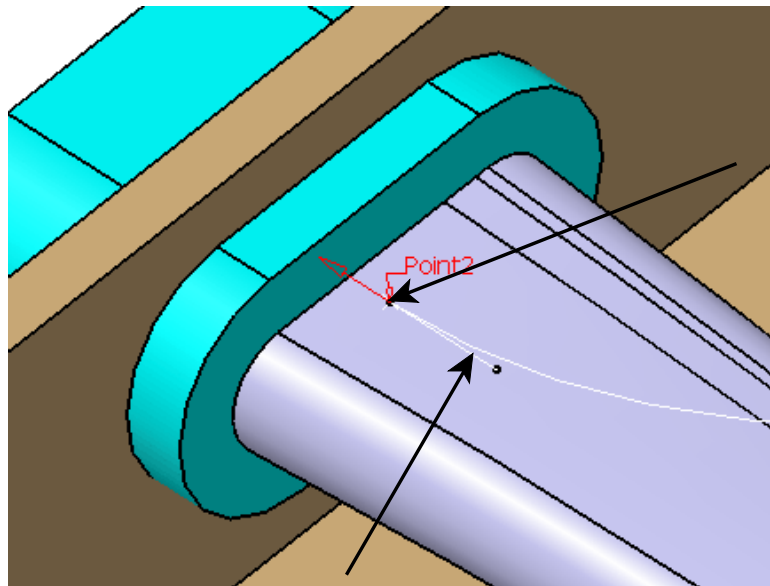
Make sure that *Distance on curve* is selected and set the *Length* to be 1.5 inches and select *OK*. The point should appear as shown.



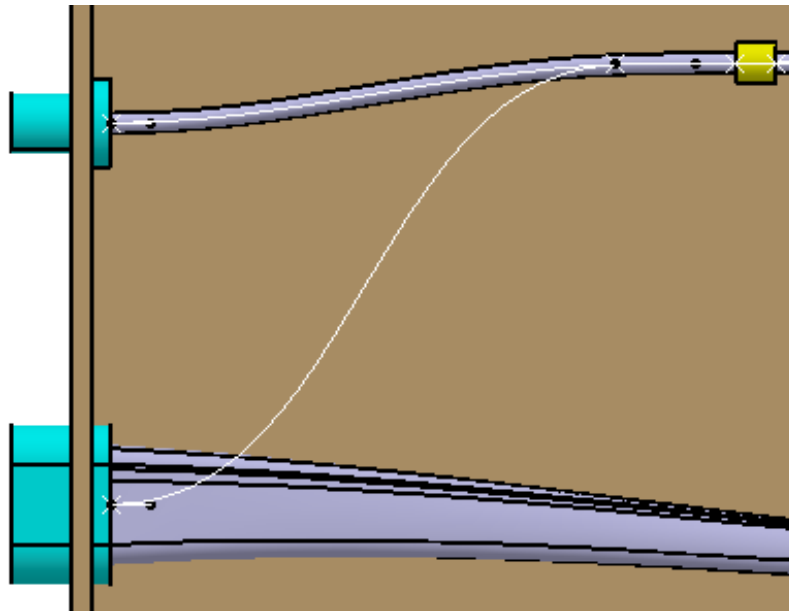
Select the **Spline** icon and select the point you just created and the spline to define the start point of the new spline and the tangency direction. Be sure the arrow is pointing as shown below.



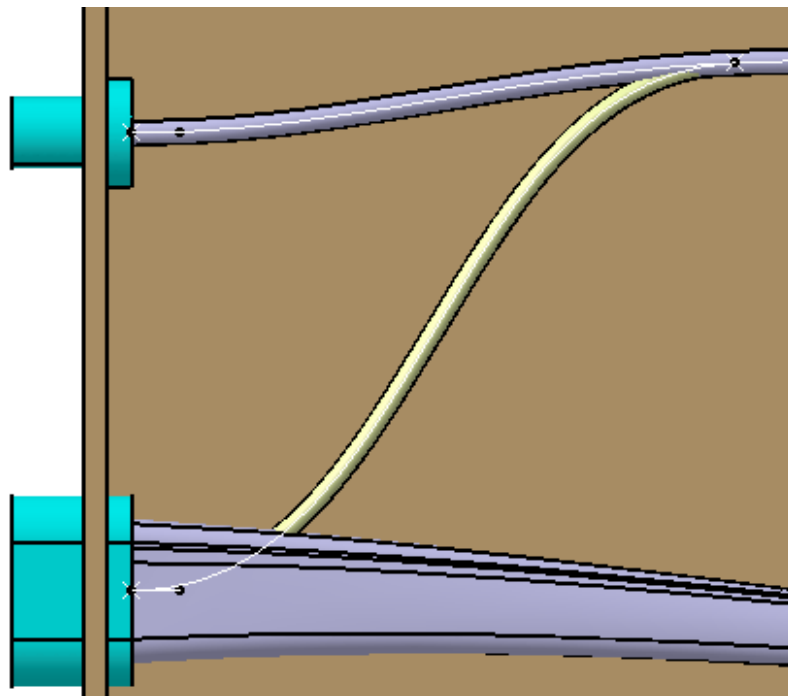
Select the point and line shown below to define the end point and tangency direction of the new spline. Be sure the arrow matches the picture.



Select **OK**. The spline should appear as shown.



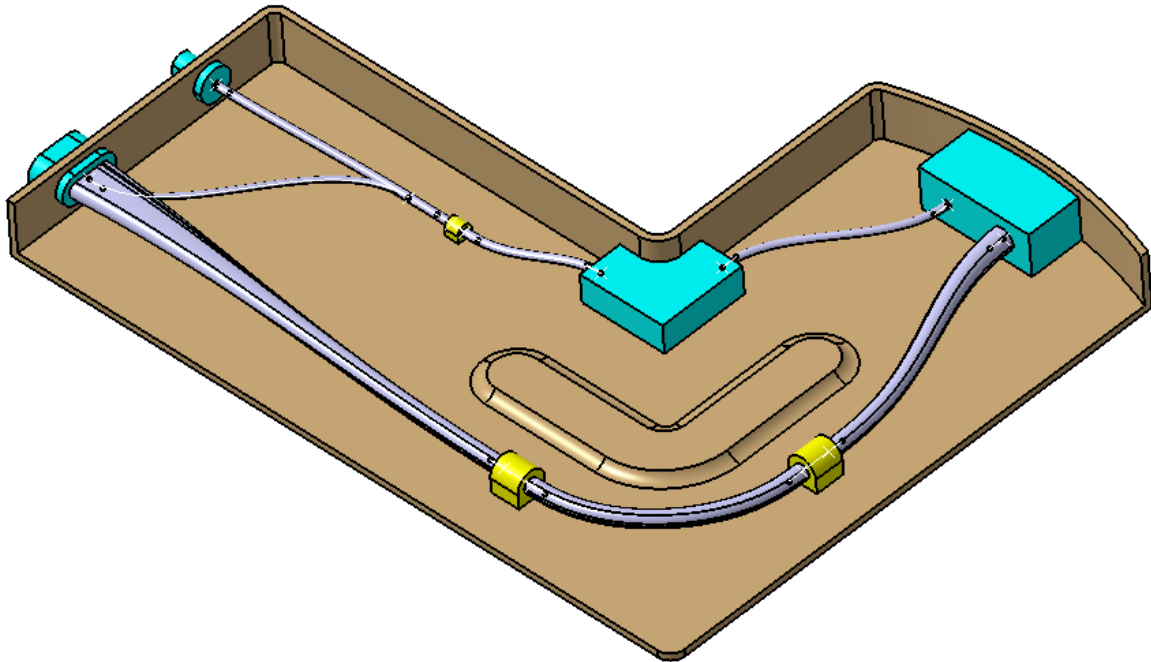
Create a **0.1 inch radius circular sweep** using the spline as a center curve. The surface should appear as shown.



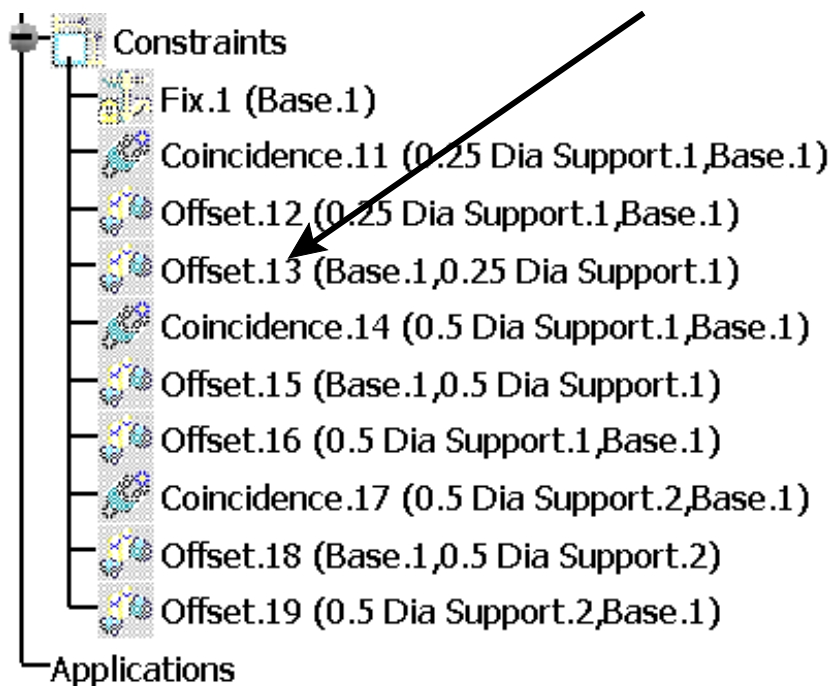
Switch to the **Part Design** workbench and close the surface.

**Hide the sweep.**

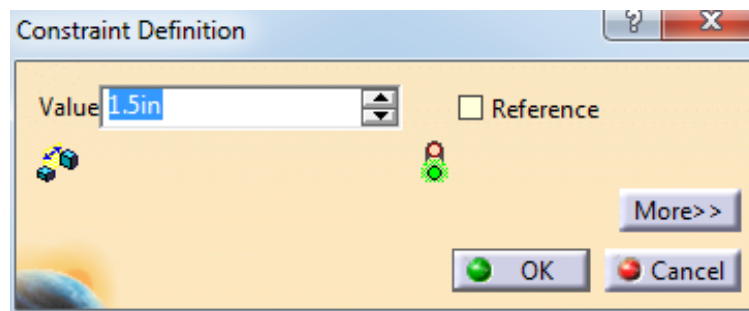
Double select on *Pathway Reservation* in the specification tree to activate the assembly. The model should appear as shown.



Double select on the *Offset.13* constraint in the specification tree.

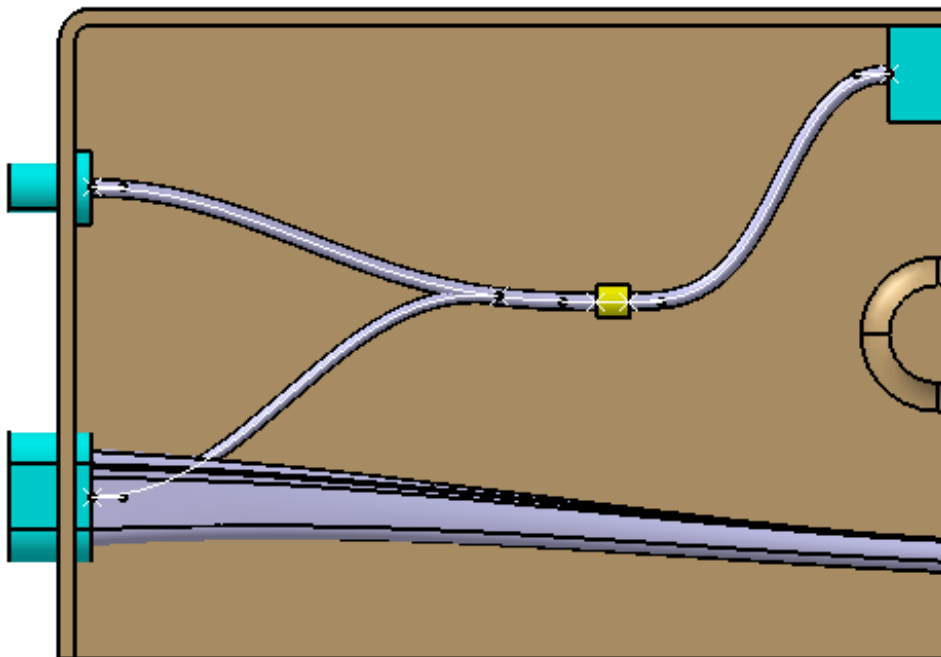


The *Constraint Definition* window appears.

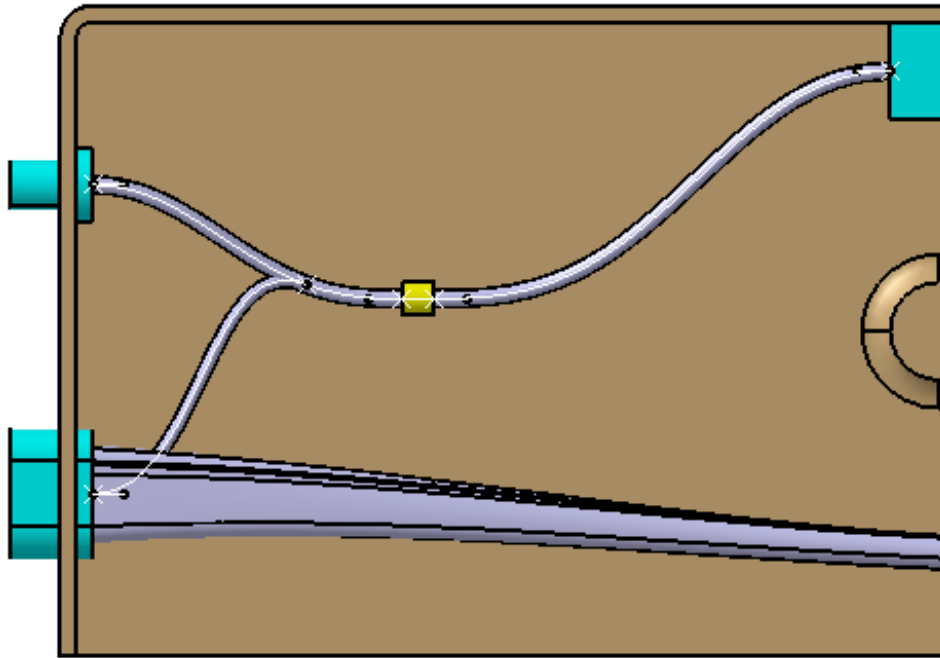


Change the *Value* to be 4.0 and select *OK*.

Update the assembly if it does not update automatically. Notice that the Support2 moves closer to the middle of the Base and all of the paths adjust accordingly.



Change the *Offset.12* constraint to have a *Value* of 5 inches. Update the assembly. The model should appear as shown.




All of the paths are linked to the equipment and support models so they should adjust if any of those models are moved.

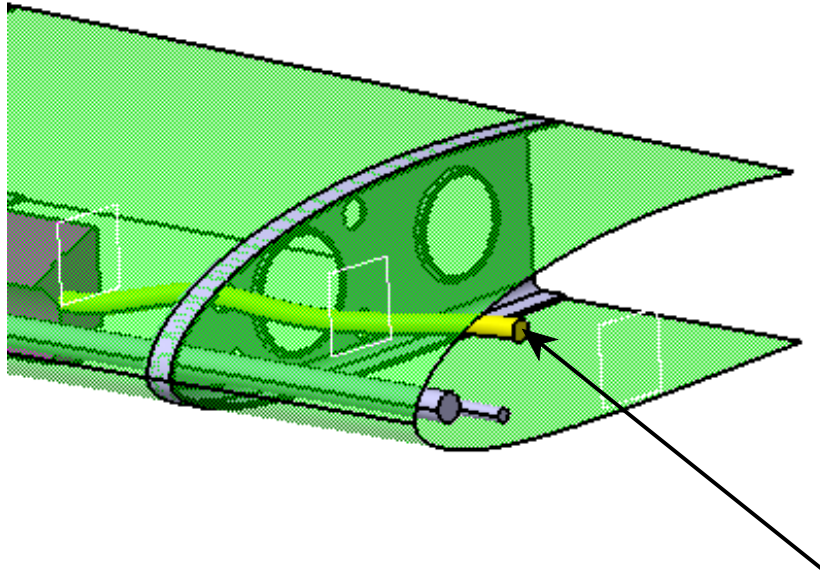
**Save and close the document.**

### 3D Annotations

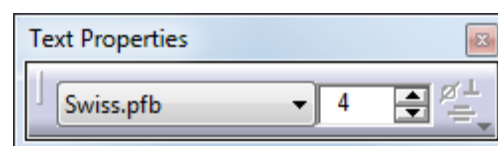
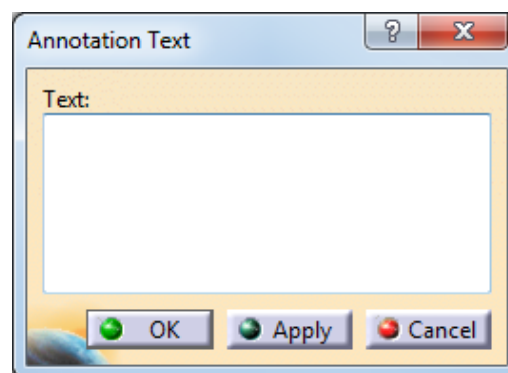
By creating a 3D annotation, you are creating an annotation that will not disappear or be removed when the model is rotated or zoomed. This type of annotation is useful for identifying specific parts in a large assembly. However, if you want to place informative markups on the drawings, you should use 2D annotated views.

Select the **3D Annotation** icon.  No windows will display until you select a part.

Select the end of the pathway as shown below.



Once you make your selection, an *Annotation Text* window appears along with a *Text Properties* window.



**Key Verify volume of tube in the *Text* field.** You can key in any amount of text that you desire.

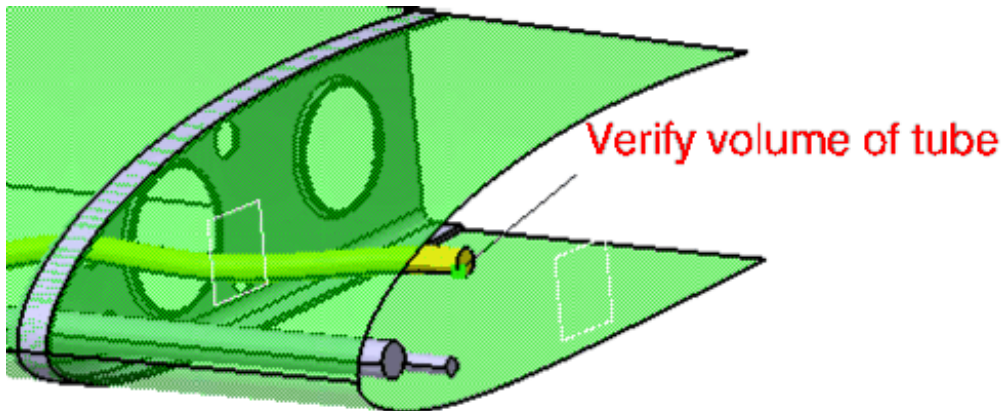
The two boxes in the *Text Properties* window will allow you to change the font and the size of the font. The drop box in the *Text Properties* window will allow you to add symbols with your text. The available symbols are shown below.



**Select *Apply*.** Notice the green dot that appears with the text where you selected the part. This is the anchor for the text.

**Press and hold the first mouse button while on the text and drag it off the part.** This will move the text to another location. You may have to rotate your model in order to select on the text.

**Select *OK*.** This creates the 3D annotation. The specification tree will have a new branch called *3D Annotations* located under the *Applications* branch.

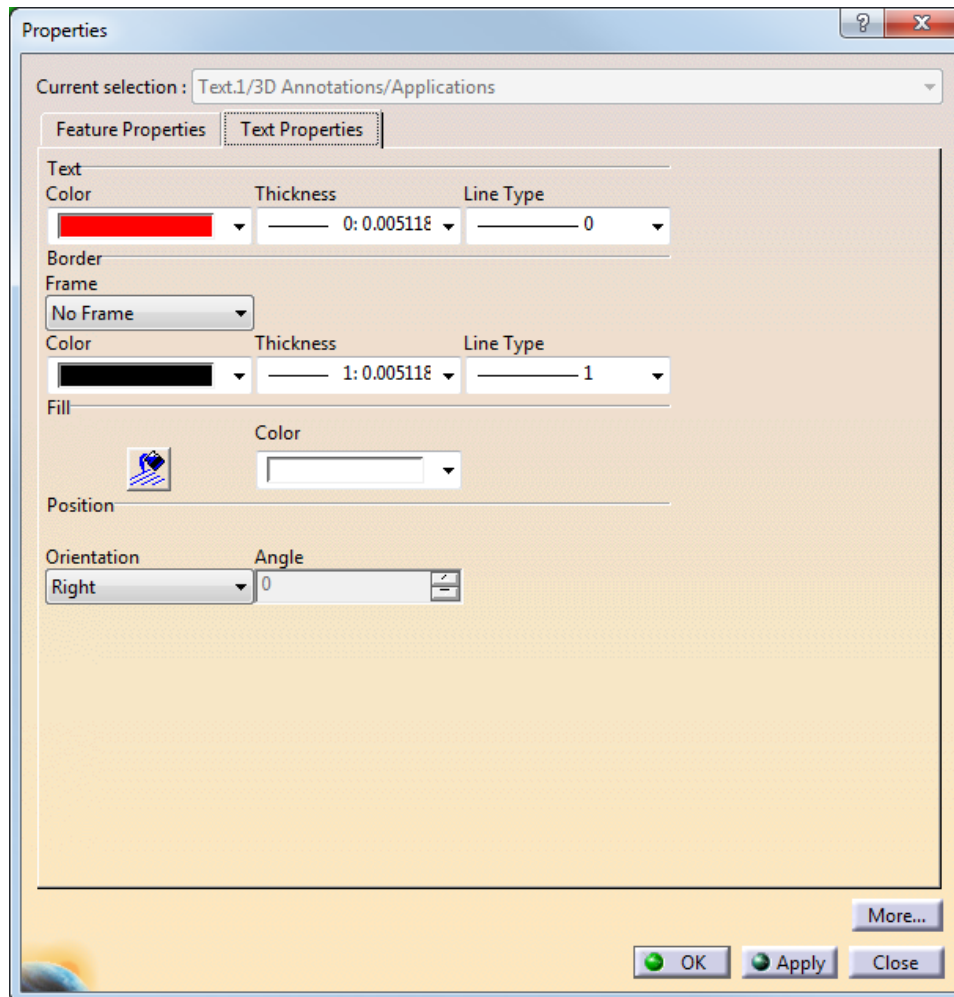


There are other features that you can perform on the text that will help make it stand out in the model.

**Using the third mouse button select on the text.** This will bring up the menu on the text.


**Select *Properties*.** This will bring up the properties of the text. There will be a few tabs along the top of the window.

**Select the *Text Properties* tab.** This tab will allow you to add borders, change the fill color and set the position of the text.



**Change the *Frame* to *Frame* and the *Border Thickness* to 3.** Select *OK* when finished. The text gets a black frame around it.

**Under the *Properties*, change the *Orientation* to *Down* and select *Apply*.** The text goes downward instead of across.

**Select the *Fill* icon and change the *Color* of the fill to yellow and select *OK*.**  The frame is filled with yellow.

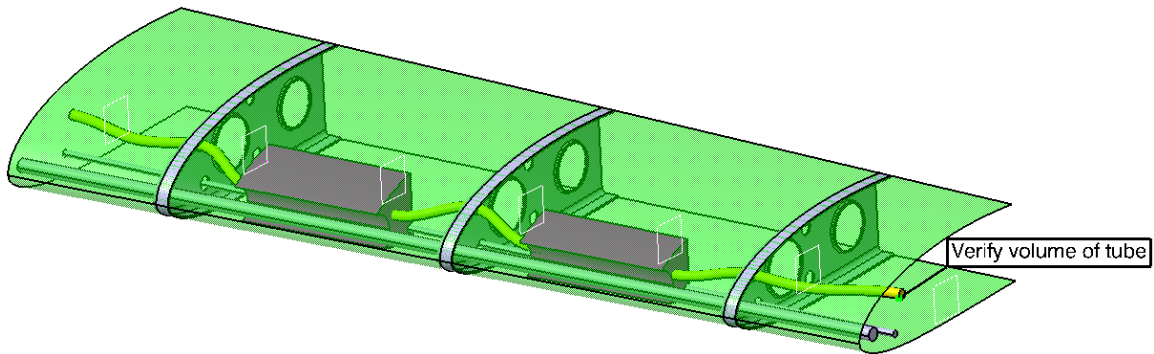
**Under *Properties*, change the *Orientation* to *Right* and turn off the fill and select *OK*.**




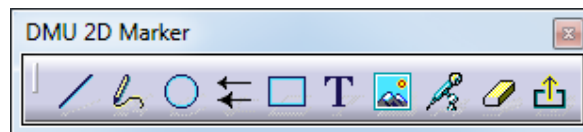
## 2D Annotated Views

In most instances, when annotations are required, a 2D annotation usually is desired. With a 2D annotation, you can create more than just text. Boxes, lines, curves, arrows, even pictures or audio comments can be added to the annotation. The key to a 2D annotation is the fact that when the model is rotated the annotation disappears. It is because of this, that you need to make sure that the model is in position before beginning the 2D annotation.

**Position the model as shown.** This will be the position for the 2D annotations.



**Select the Annotated View icon.**  After selecting this icon, a 2D annotated view is defined. There will also be an additional set of tools displayed in a *DMU 2D Marker* toolbar.



Creates a line



Allows you to freehand around the 2D annotation



Creates a circle



Creates an arrow on the annotation



Creates a square




Allows you to key text on the annotation




Allows you to insert various picture formats on the annotation plane



Inserts an audio marker on the annotation

 Erases all annotations

 Exits the annotated view

You can select the icon of the shape you wish to create. Once the icon is selected, you select where you want the shape to begin and where you want it to end.

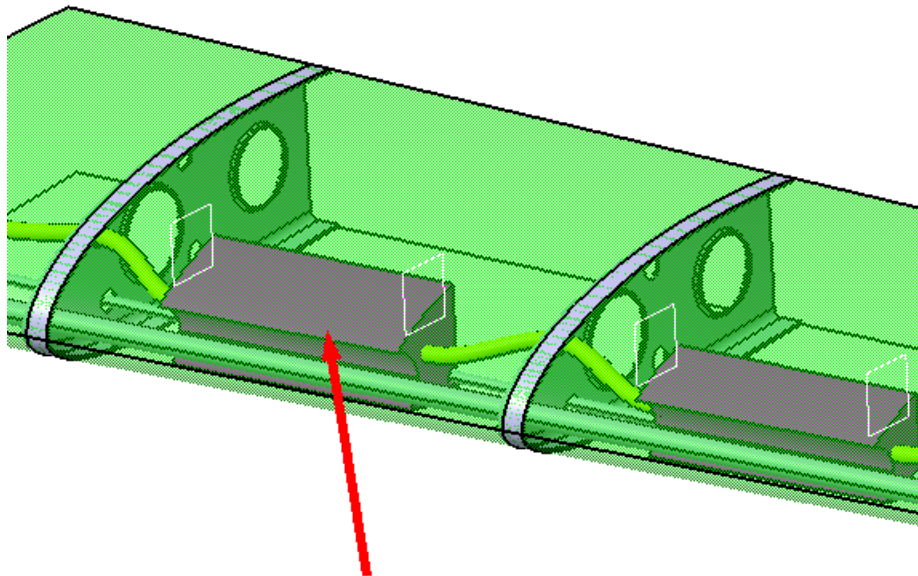
You can select the text icon and specify where you want the text to appear. The *Annotation Text* window will display, allowing you to key in text and change the font properties similar to the 3D annotation options.


To insert a picture, select the picture icon and select where you want the lower left corner of the picture to appear. This will bring up a *Select Picture File* window allowing you to specify the picture you want to insert into the annotation. You can insert tiff, jpg, bmp and rgb image files into the annotated view.

To insert an audio marker, select the audio marker icon. This will bring up a *Select audio File* window. You can then define the file that the audio annotation is going to be recorded into.

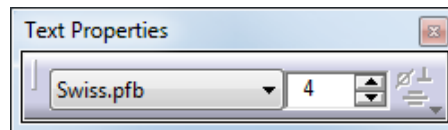
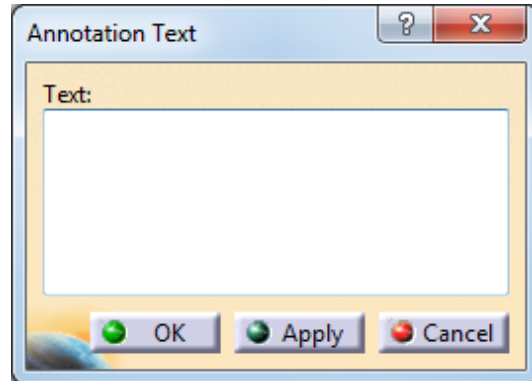
**Select the Draw Arrow icon.**  This will allow you to create an arrow.

**Select and hold the first mouse button away from the model at the base of the arrow shown below, then drag towards the Equip1 part, releasing on top of the part.** This will create the arrow shown below.

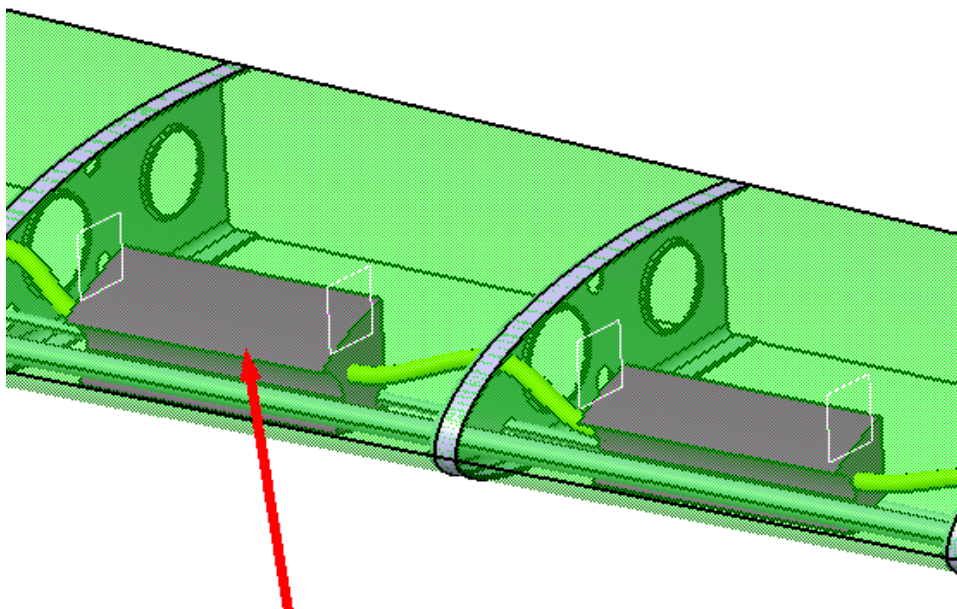


Select the **Add Annotation Text** icon.  This will allow you to define text in the 2D annotation.

Select **past the end of the arrow in space**. This defines where the top left side of the text will be. An *Annotation Text* window appears along with a *Text Properties* window.

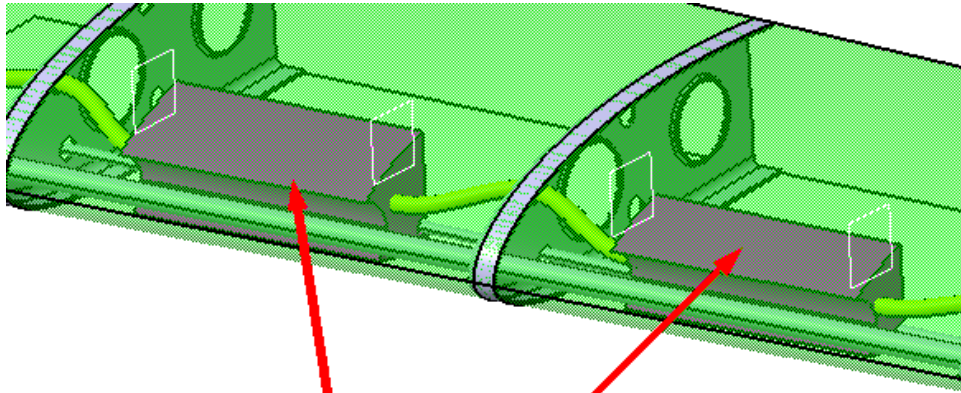


Key Verify all instrumentation will fit within volumes in the window and select **OK**. Any amount of text can be keyed into the window. You can move the text by selecting and dragging the text as you need.




**Verify all instrumentation  
will fit within volumes**

Go ahead and draw another arrow and a box as shown below.



Verify all instrumentation  
will fit within volumes

Select the **Exit** icon in the toolbar.  This will exit you out of the 2D view. Notice that the annotations that were created in the 2D view disappear. The view is located in the *Annotated Views* branch under the *Applications* branch. To view it again, you need to double select on the view in the specification tree.

Save and close the document.