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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

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

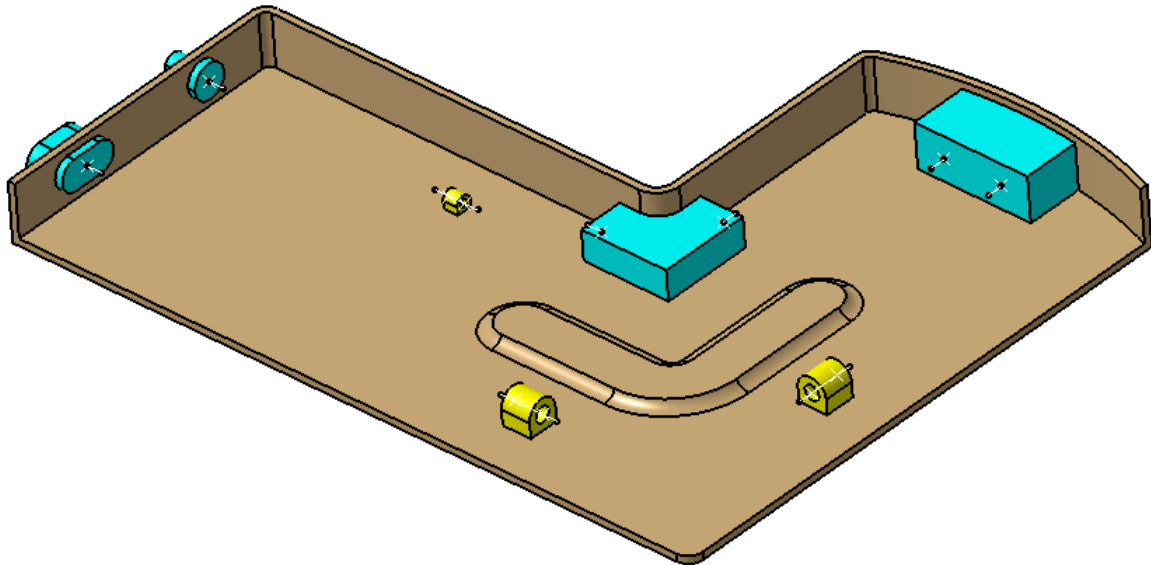
Creating an electrical reservation involves creating shapes and areas that will act as a keep-out zone or an area to avoid for other disciplines. Although CATIA V5 has a product for Systems Space Reservations, you are limited to creating reservations that are only simple shapes. Many times you will have 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, as well as 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.

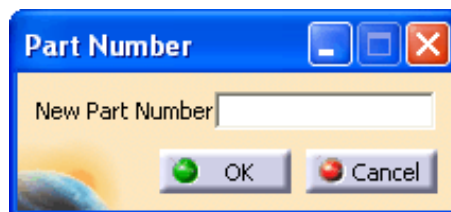
Open the Pathway Reservation document from the *Pathway Reservation* directory.
The model appears as shown.



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.

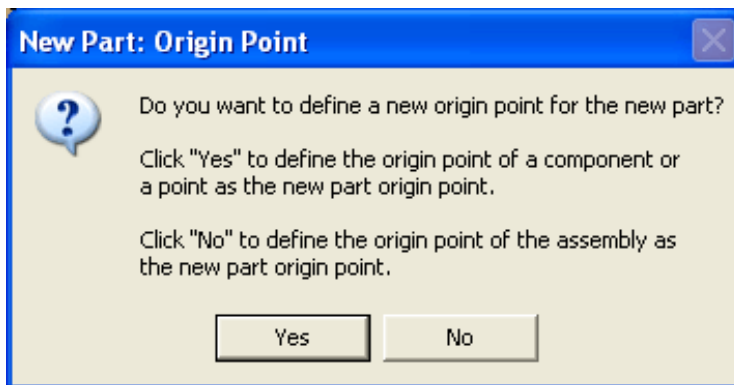
Be sure that you are in the Assembly Design workbench.

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

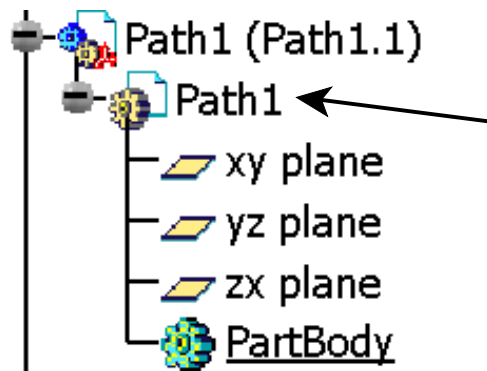


Key in Path1 for the *New Part Number* and select *OK*.

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.



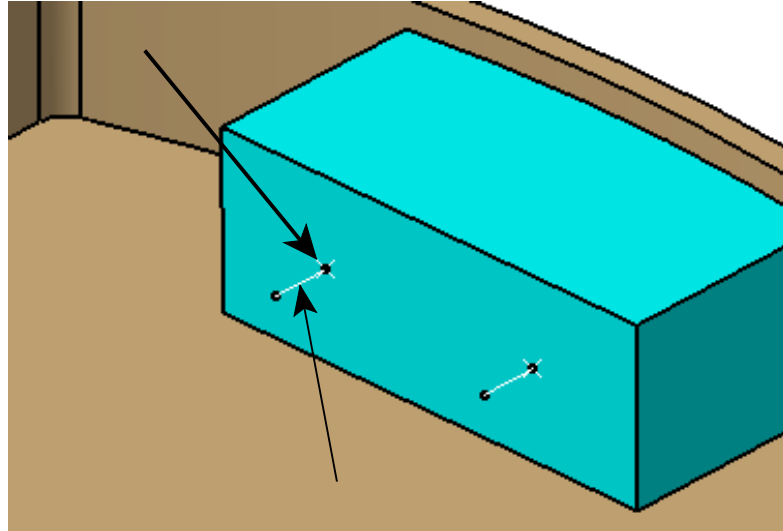
Double select the *Path1* document as shown above to activate the part. This should switch you to the Part Design workbench. You are going to build this part within the context of the assembly.

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

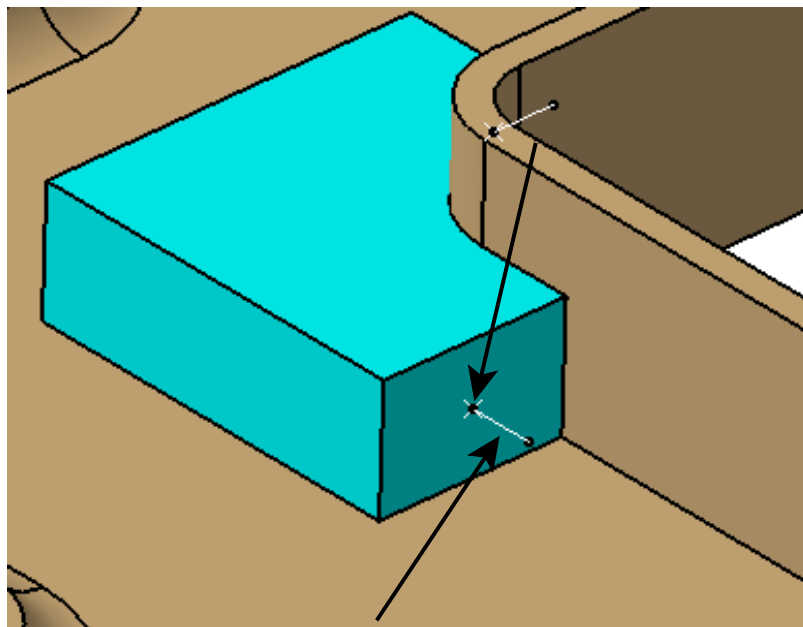
Hide the reference planes of the part. You will not need the reference planes.

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

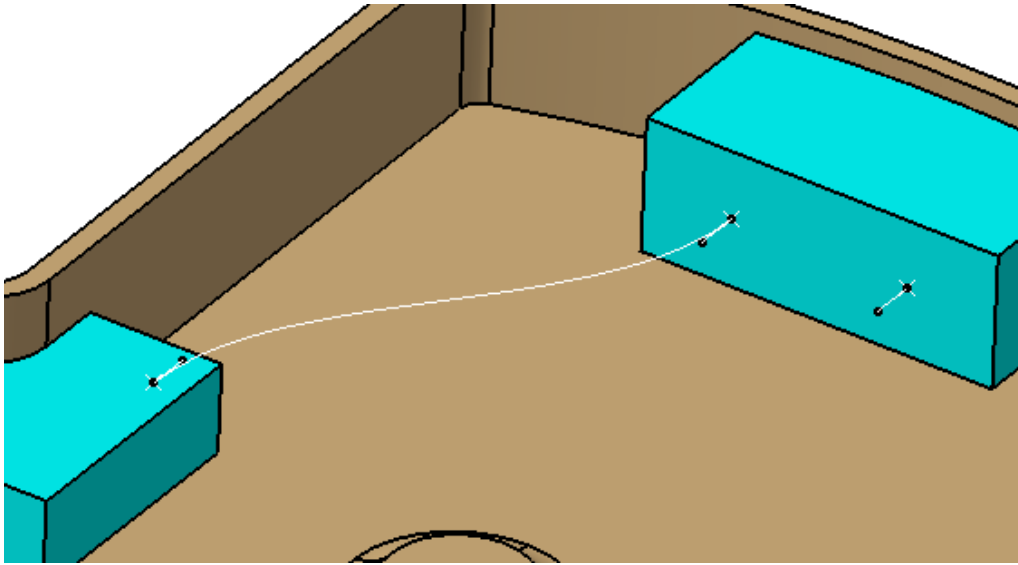
Select the point and line as shown below to define the start location and tangent direction of the spline. Be sure that the tangent direction arrow is pointing out of the box.



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.



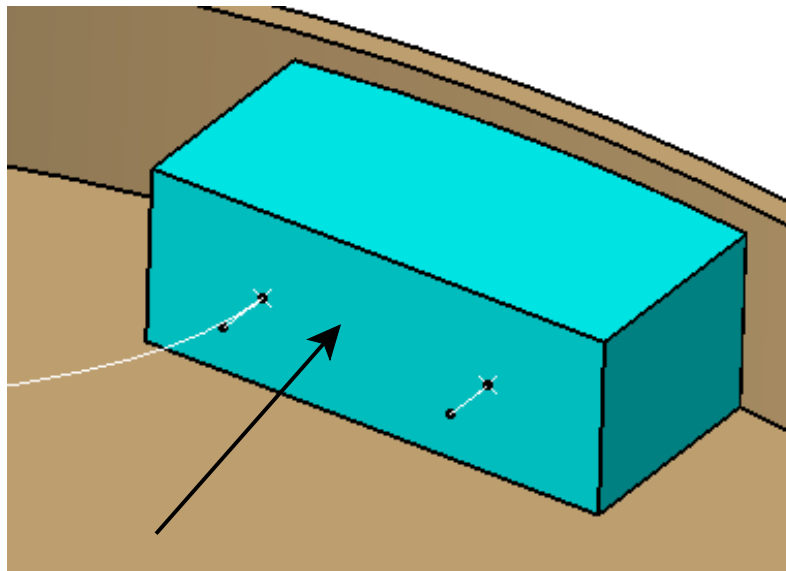
Select **OK**. The spline should appear as shown.



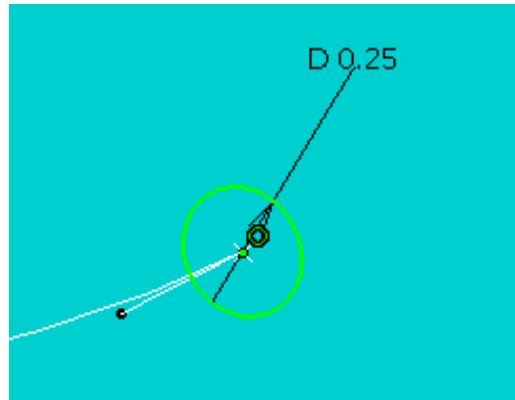
You will create a few different pathways so we will look at some different options for doing each path.

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.

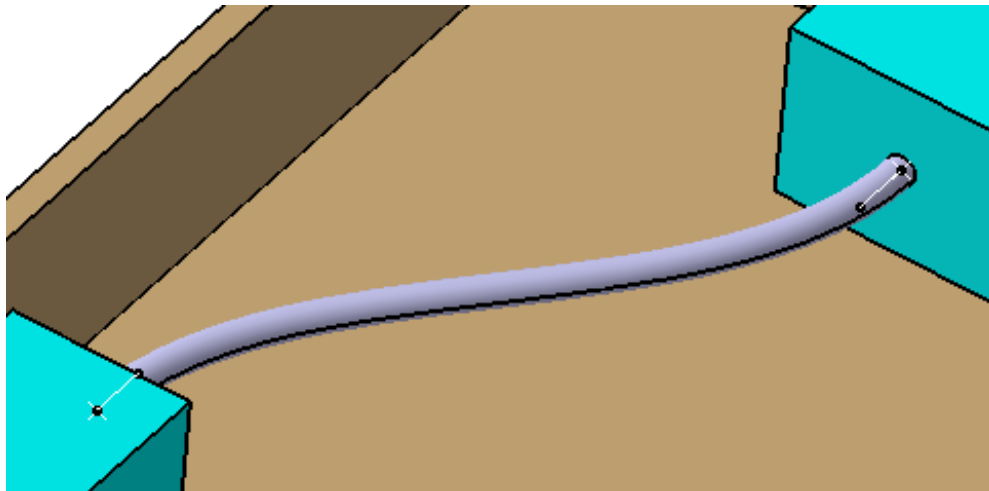


Exit the sketch.

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.

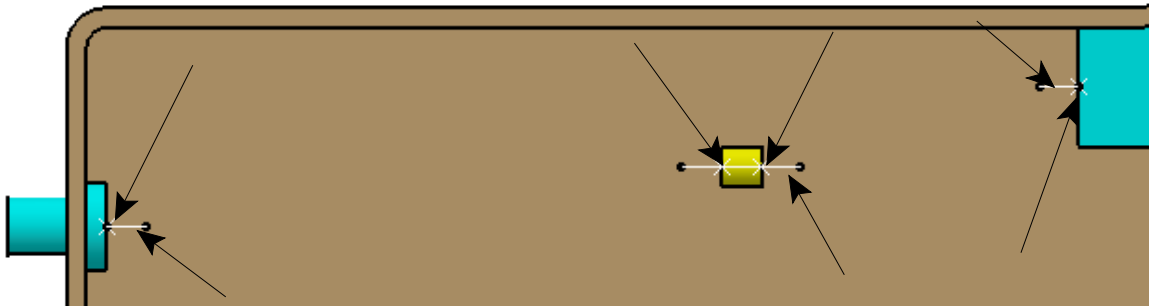
Use *Save Management* to save the assembly and all of the parts in a folder called Pathway Reservation Assembly in your directory.

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

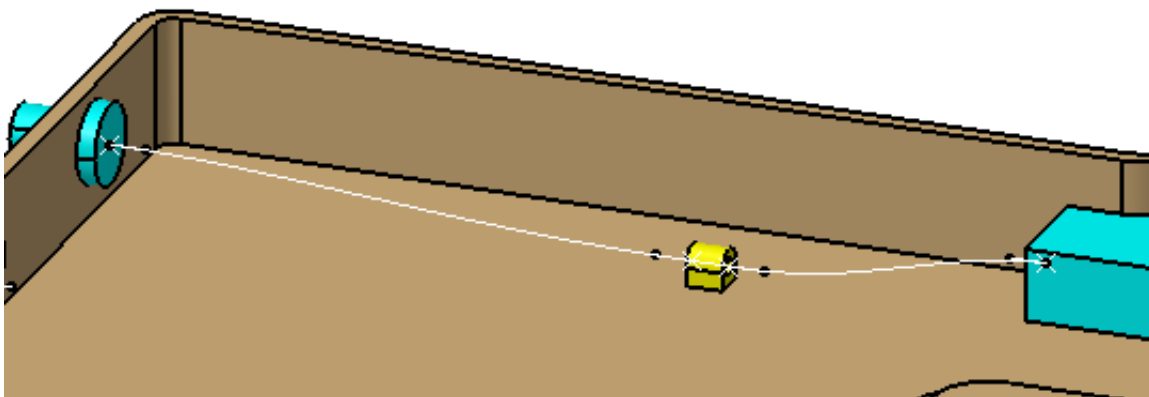
Activate the *Path2* document and hide the reference planes of the part.

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.

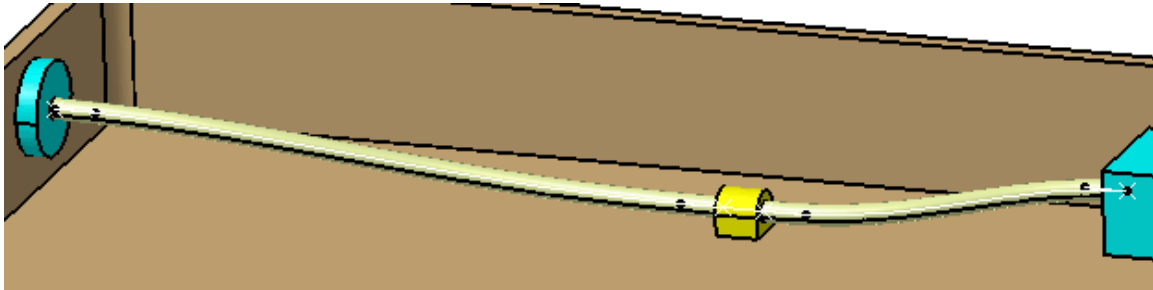
Select the Sweep icon.  The *Swept Surface Definition* window appears.

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.

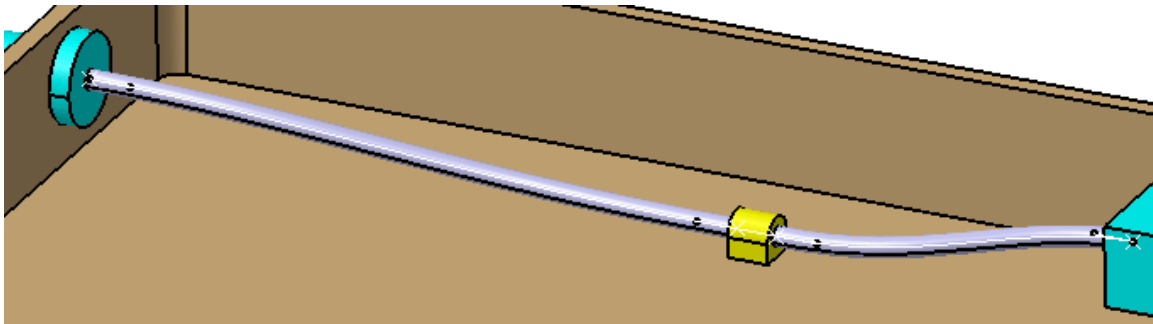


Switch to the **Part Design** workbench.

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.

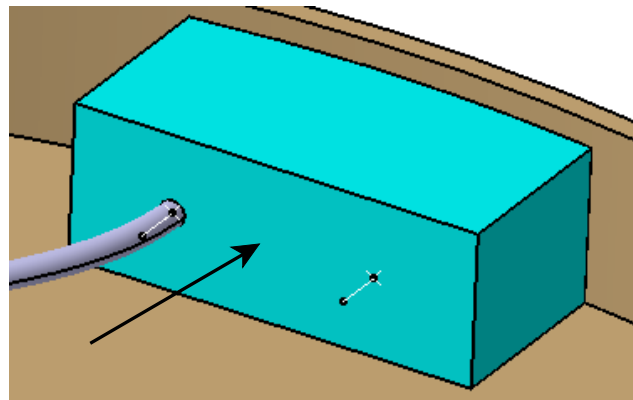


Double select on *Pathway Reservation* in the specification tree to return to the product level.

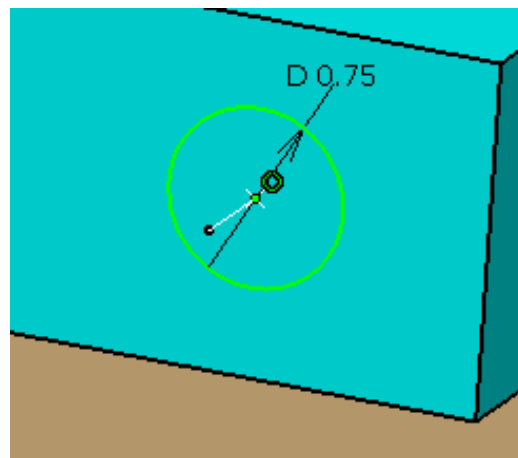
Create a new part called Path3 in the same manner as before.

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.

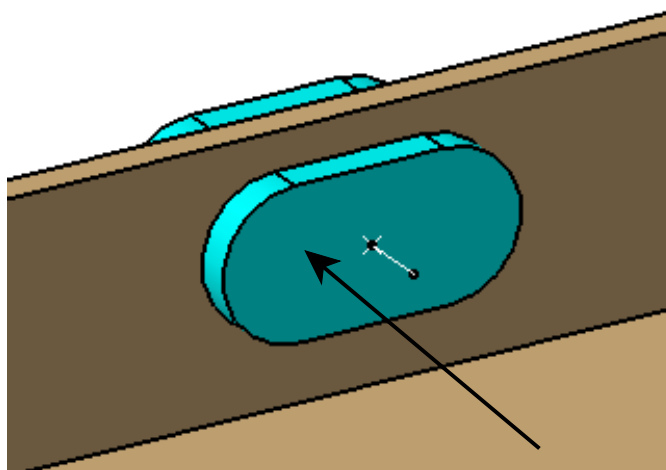


Draw and constrain a circle as shown below. The center point of the circle should be constrained to the point.



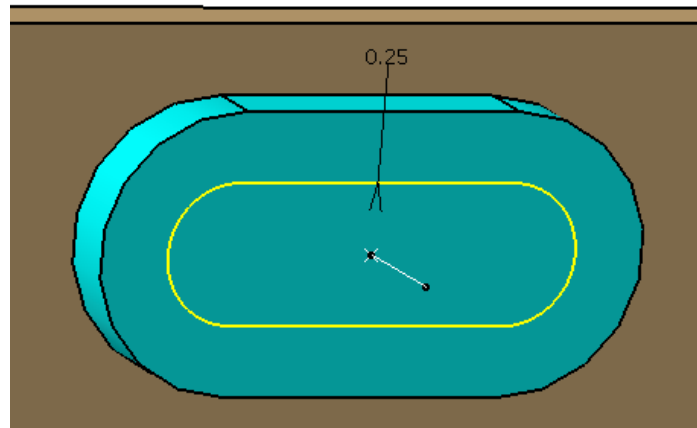
Exit the sketch.

Create another sketch on the face shown below.



Select the Offset icon and select the same face again. 

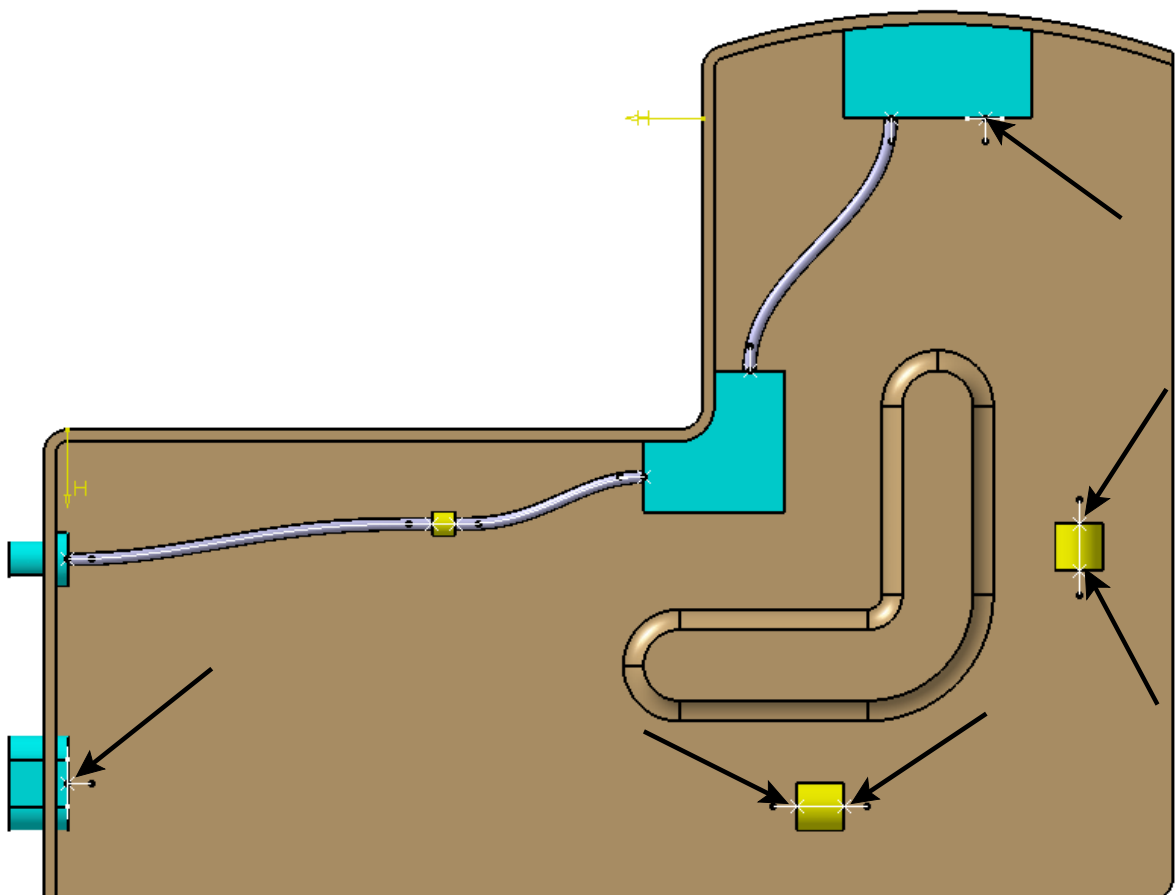
Set the offset to be 0.25 inches. The sketch should appear as shown.



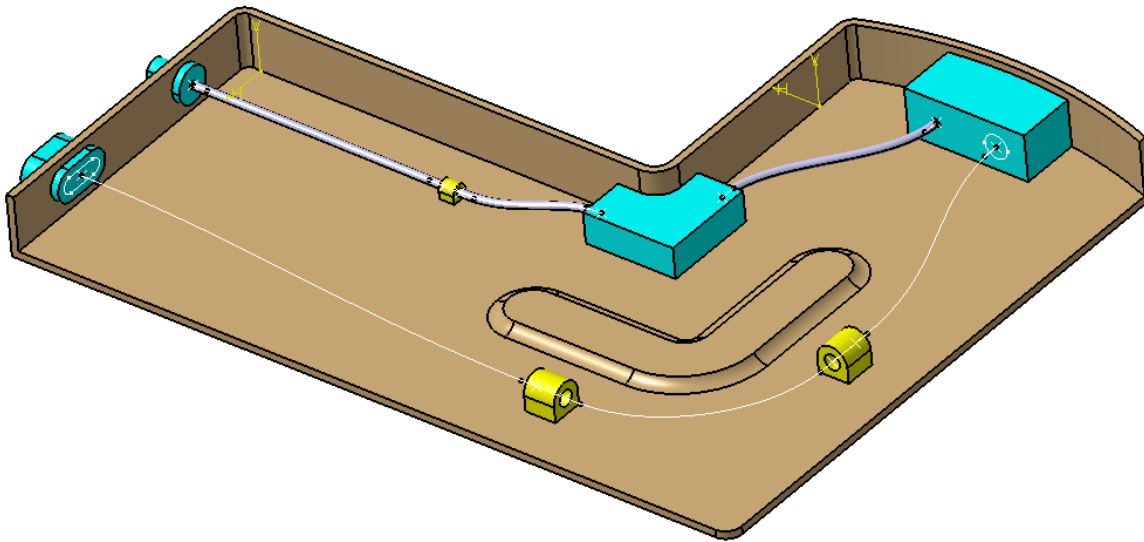
Exit the sketch.

Switch to the Generative Shape Design workbench.

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



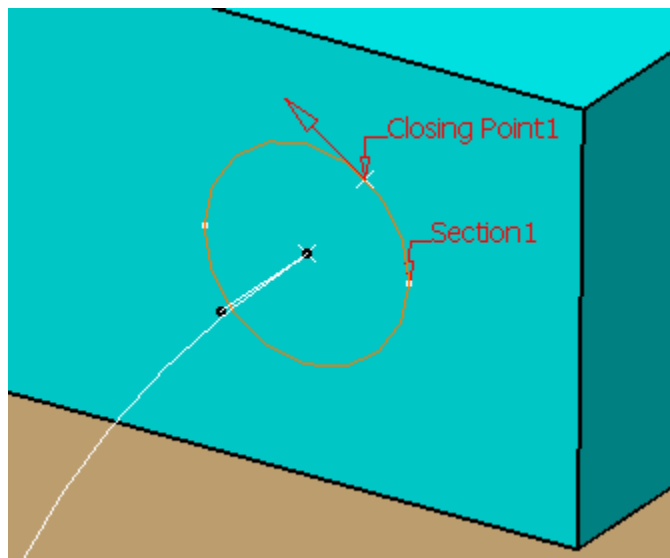
The spline should appear as shown.



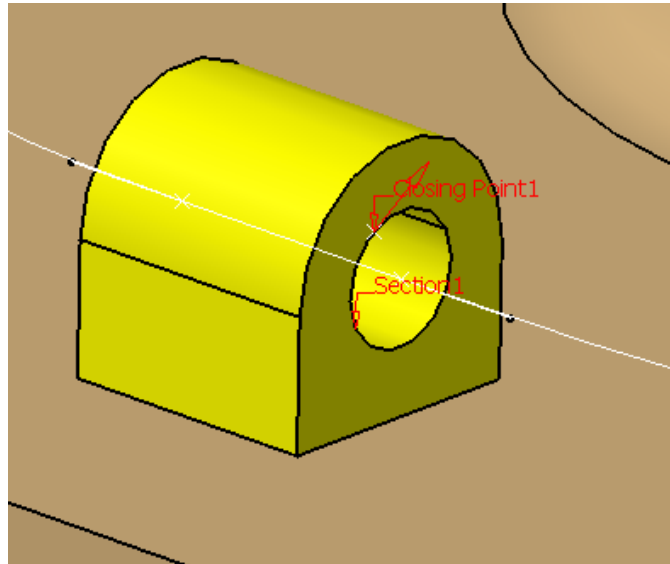
Switch back to the Part Design workbench.

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.

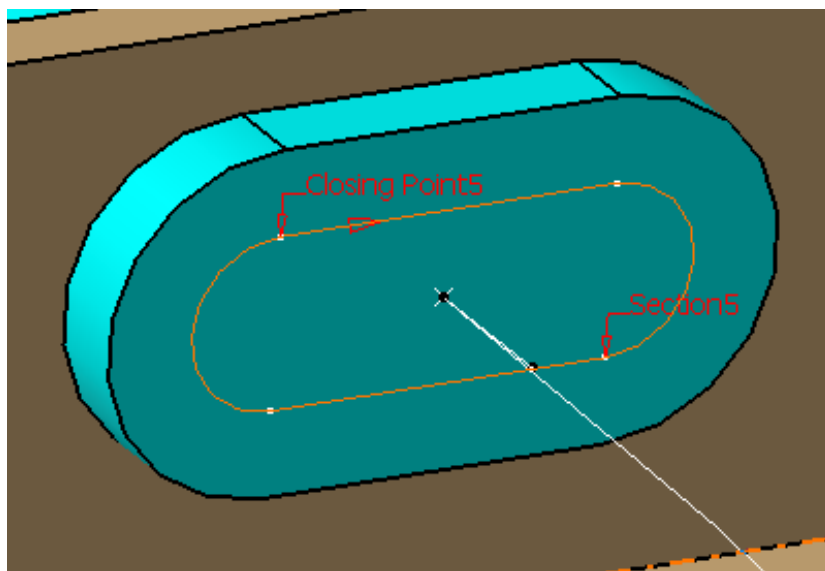


Select the circular edge as shown to define the second profile. Make sure the *Closing Point* arrow matches the picture.



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.

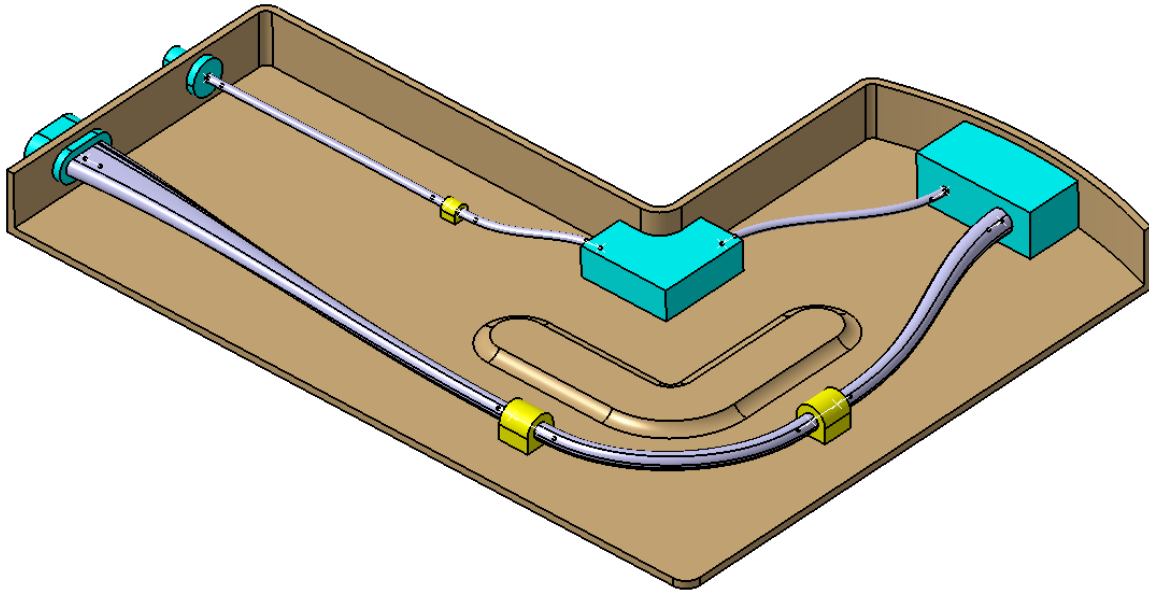
Select the elongated hole sketch to define the last profile for the multi-sections solid. Again be sure the arrow is pointing as shown.



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.

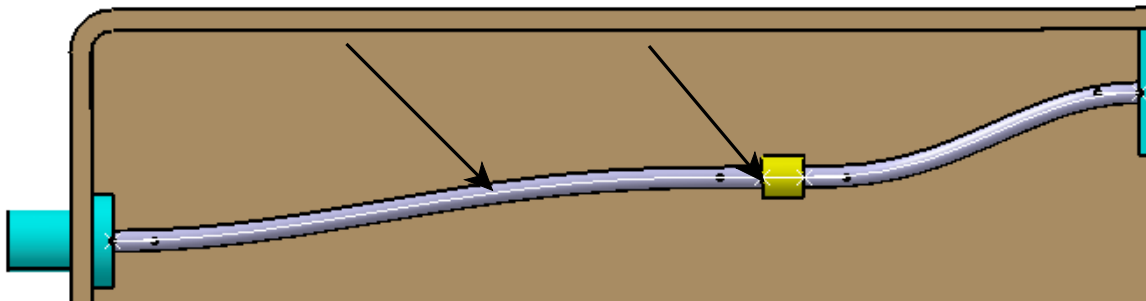
Activate the part and hide the reference planes.

Switch to the Generative Shape Design workbench.

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

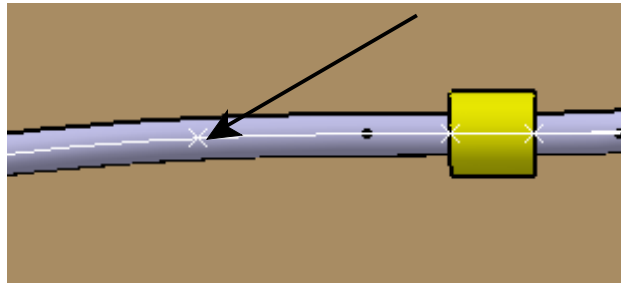
Change the *Point type* to *On curve*.

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

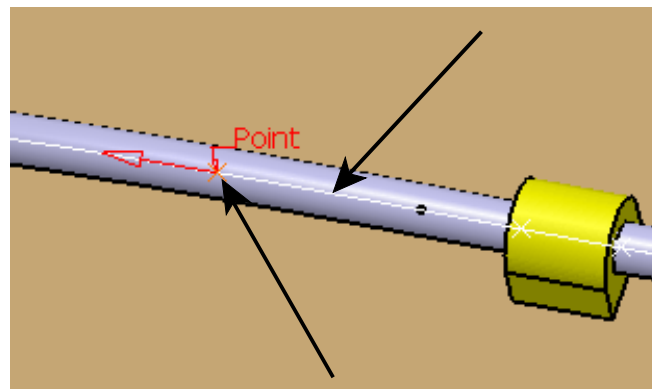


Select in the *Reference Point* box in the window and select the point as shown above to define the *Reference Point*.

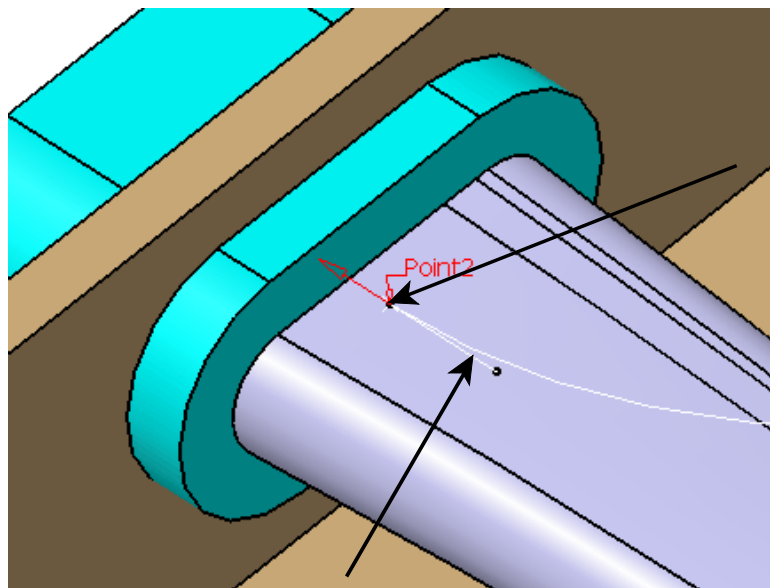
Make sure that *Distance on curve* is selected and set the *Length* to be 1.5 inches and select *OK*. The point should appear as shown.



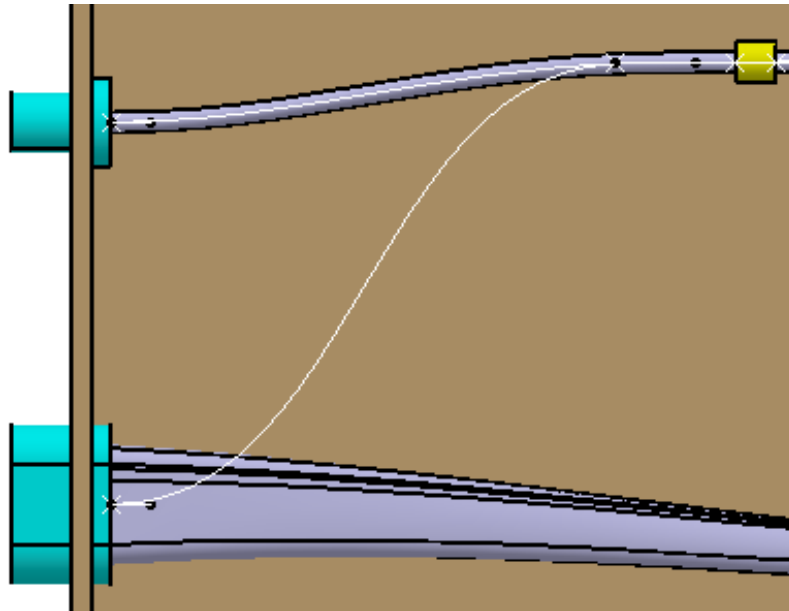
Select the Spline icon and select the point you just created and the spline to define the start point of the new spline and the tangency direction. Be sure the arrow is pointing as shown below.



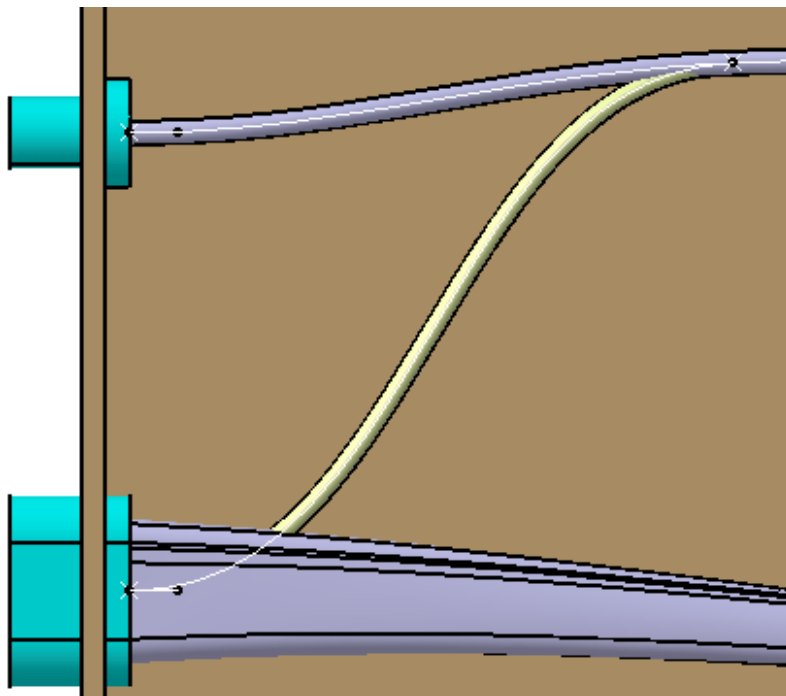
Select the point and line shown below to define the end point and tangency direction of the new spline. Be sure the arrow matches the picture.



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



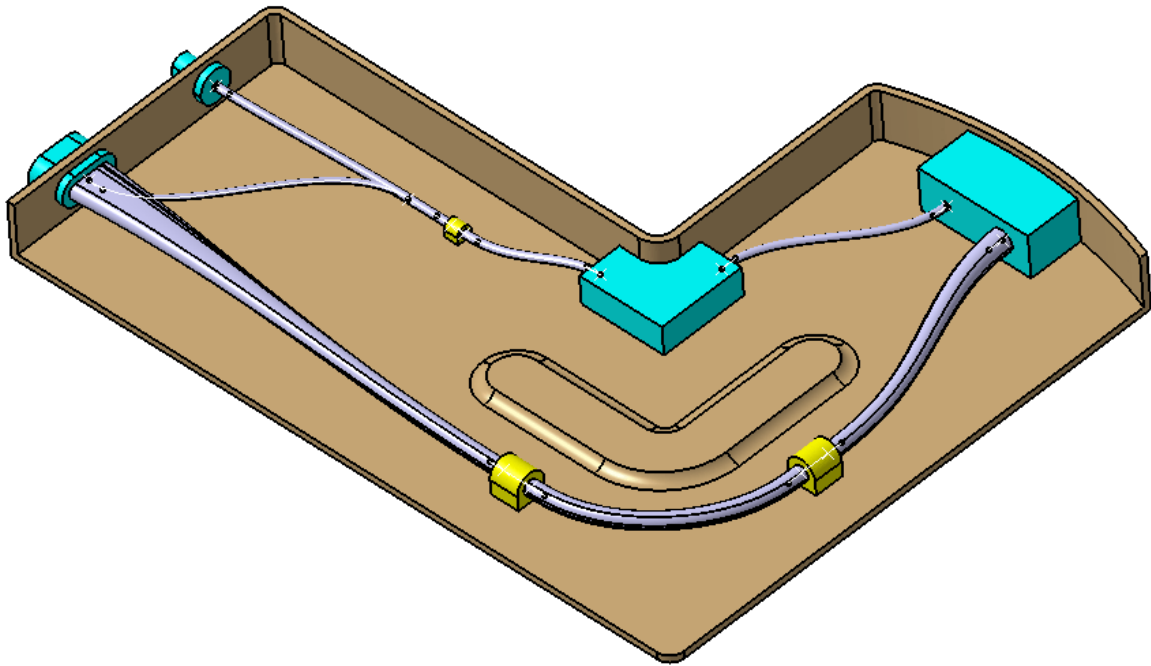
Create a 0.1 inch radius circular sweep using the spline as a center curve. The surface should appear as shown.



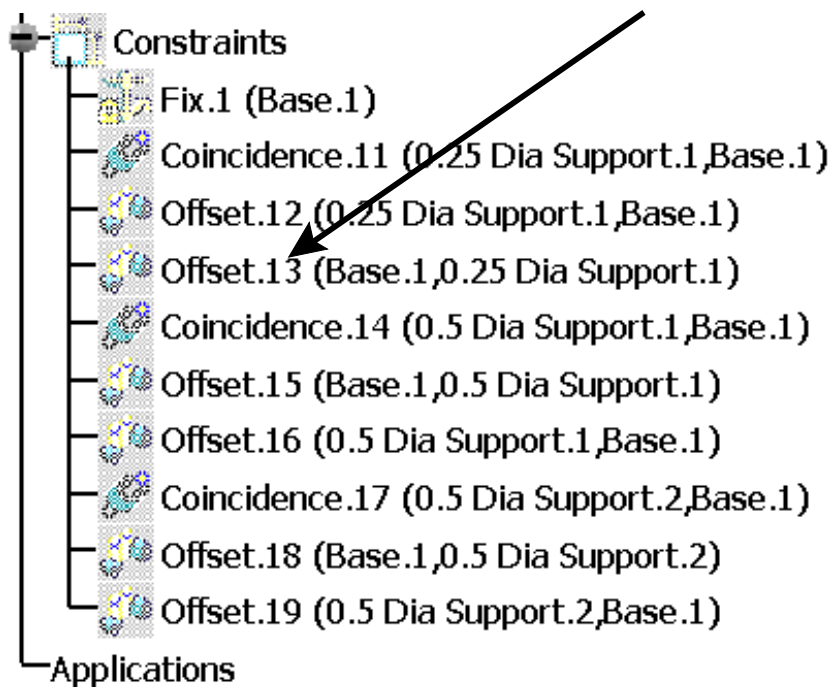
Switch to the Part Design workbench and close the surface.

Hide the sweep.

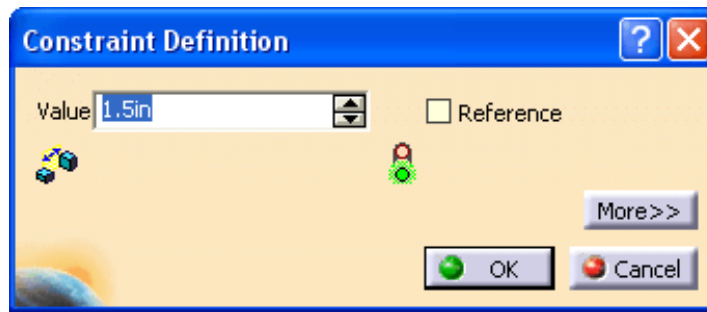
Double select on *Pathway Reservation* in the specification tree to activate the assembly. The model should appear as shown.



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

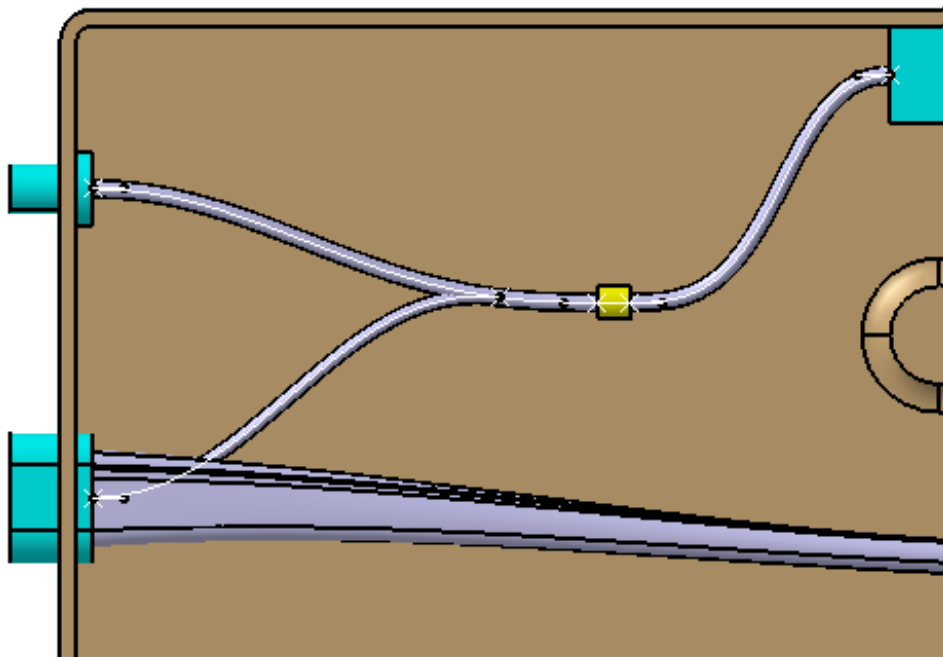


The *Constraint Definition* window appears.

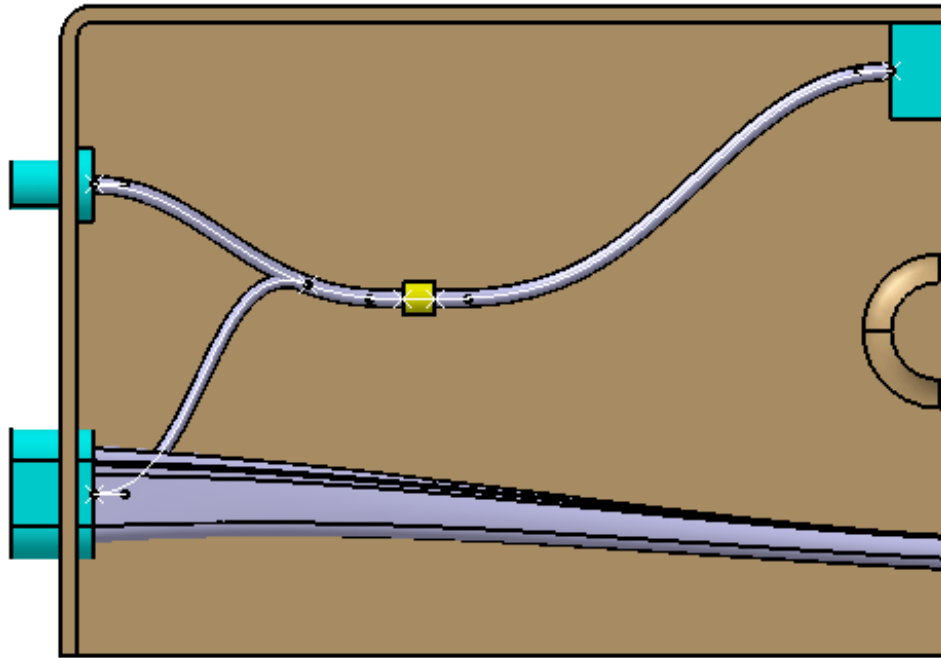


Change the *Value* to be 4.0 and select *OK*.

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.