Make a hose.

Today's Lesson is Sponsored by Raytheon

Contents

Changing the View of a Design

- if you don't see a grid in the Fusion 360 window, as shown below, click on **Grid and Snaps** and check **Layout Grid**. Displaying the *Layout Grid* is a matter of preference. When designing for 3D printing, it can be used to represent the *build plate*.

- click on the **Orbit** tool and click somewhere on the **Grid** to practice rotating and changing the angle of the view.
- click on the **Pan** tool and then on the **Grid** to practice moving the view laterally.
- after using the *Orbit* or *Pan* tool one must press the **Esc key** to exit that mode.
- use the **Mouse Wheel** to practice Zooming in and out.

Here is a close-up of the View Cube at the top right of the window.

- click on the **View Cube** and move the cube while holding the mouse button down. This is another way to rotate the view.
- click on the Top of the View Cube and note how the view just jumped to a Top View.

The View Cube now resembles that on the right.

- click on the **Curved Arrows** at the upper right of the View Cube and practice Rotating the View.
- click on the **Arrows** at the sides of the View Cube to practice jumping to various Views.
- click on the **Home** icon to the upper left of the View Cube. This can always be used to reset the view to the Home View
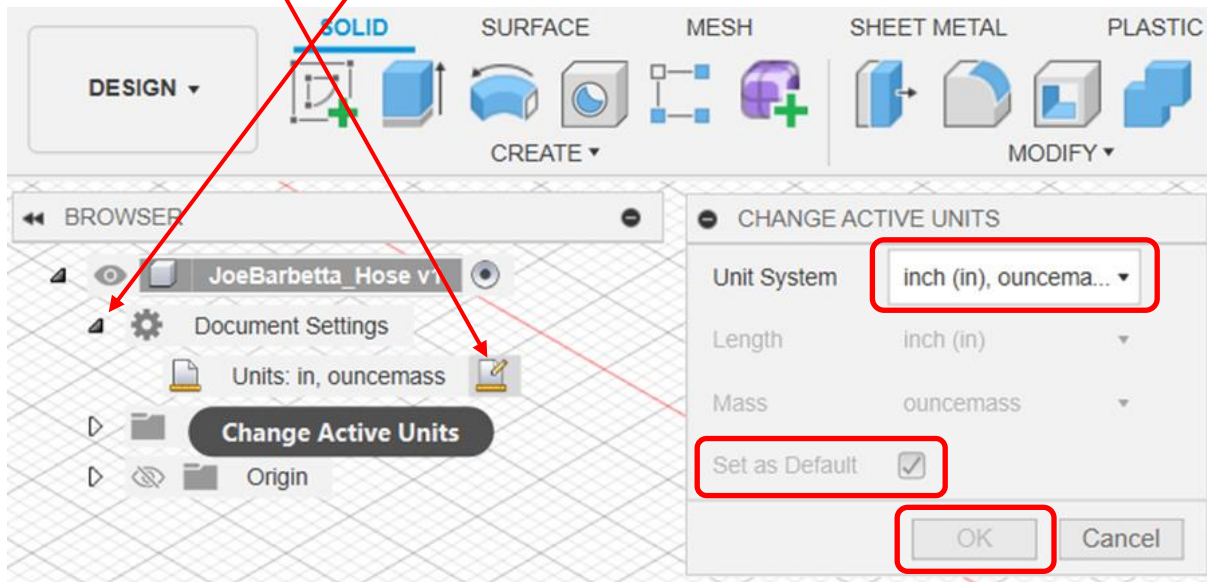
Starting a Design in Fusion (START HERE)

- open **Fusion**. If there is no icon on the Desktop, use the Windows search (magnifying glass icon) and type fusion
- from top **File** icon select **Save** and name the file.
   Use your name followed by **_Hose** e.g. **JoeBarbetta_Hose** (note the use of the underscore)

Note that by default Fusion saves your project to "the cloud", which are the servers managed by AutoDesk. When you log into Fusion on a different computer, your projects will be available.

As you work you may want to occasionally save your work in case Fusion crashes or we lose power.

- in the left **"BROWSER"** click the **arrow next to Document Settings**
- click on the **edit icon** that appears to the right when you hover over **Units**
- ensure **Active Units** are set to **Units: in, ouncemass** and click **OK**. You can also enable **Set as Default** if it is not grayed out.

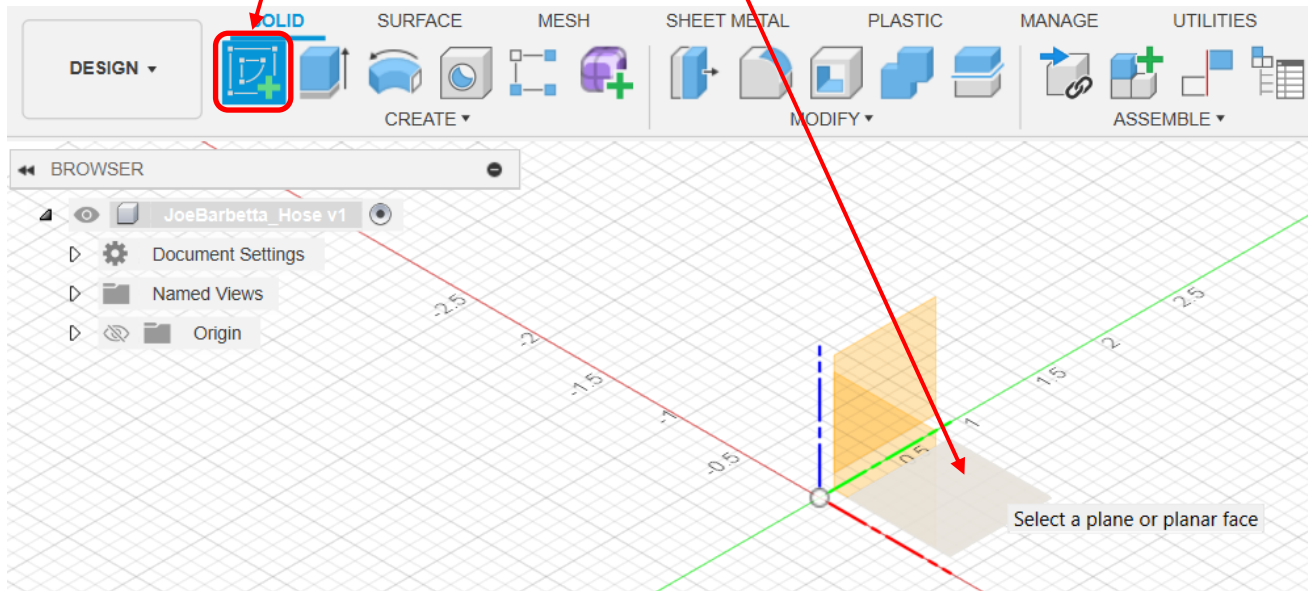Note that the default units are in mm, which we just changed to inches.

Did you know that the default units have changed over the years? The earliest version used cubits as the default unit.

Creating the First Sketch

Note that a Fusion expert may tell you to create a Component first. Just say **"Dude. I'm just making a hose."**

- select the top **Create Sketch** tool and click on the **bottom rhombus** to select the X-Y Plane.
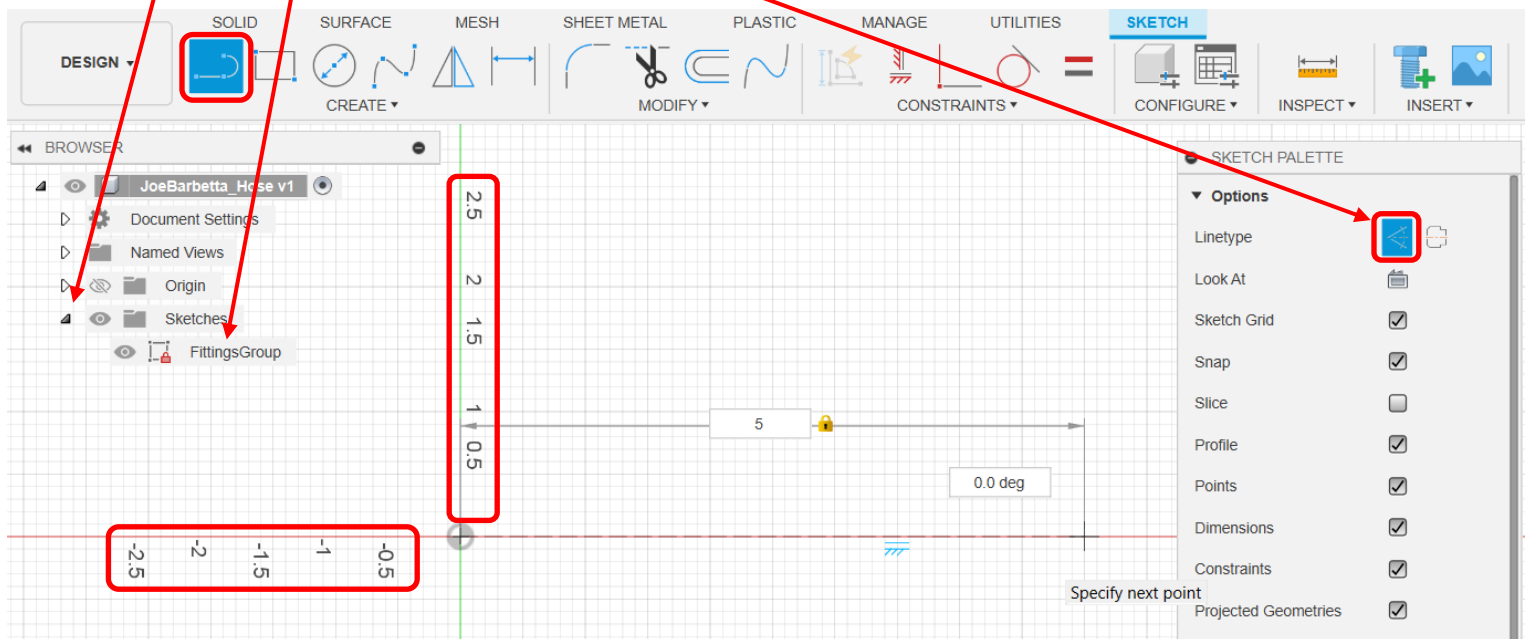If a tool can't be found, one can always look in the **CREATE** and **MODIFY** menus for it.

- zoom in as shown below. The scale labels can give an idea of how far one is zoomed in. The **View Cube** should indicate you are sketching on the **Top X-Y Plane**.
- click on the **arrow** next to the Sketches folder to open it
- right click on the Sketch name, select **Rename** from the menu and rename the Sketch to **FittingsGroup**
- select the **Line** tool and click on the **Construction icon** for **Linetype** to highlight it blue.
- click on the **Origin**, extend the line **to the right**, type **5**, and press the **Enter key**
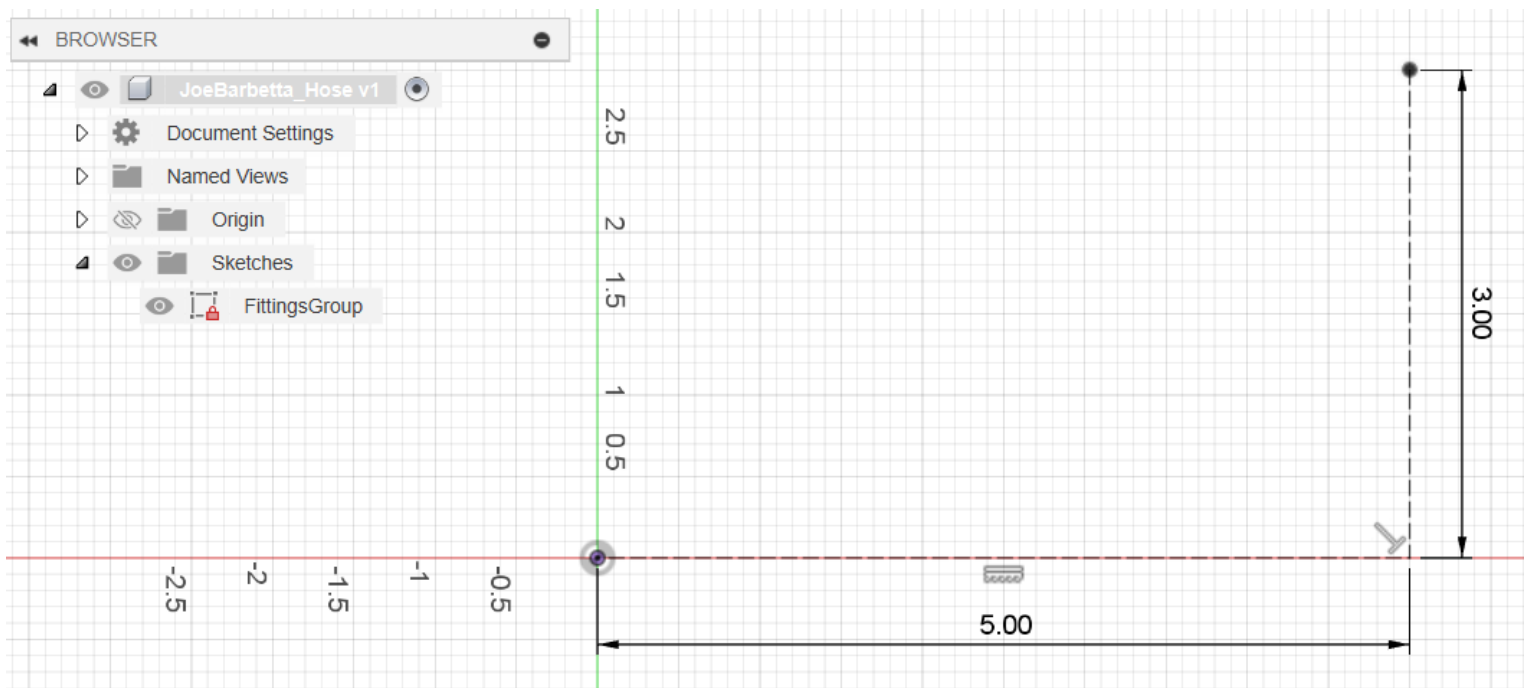
- select the **Line** tool again and click on the **end of the last line**, extend the **line upward**, type **3**, and press the **Enter key**.
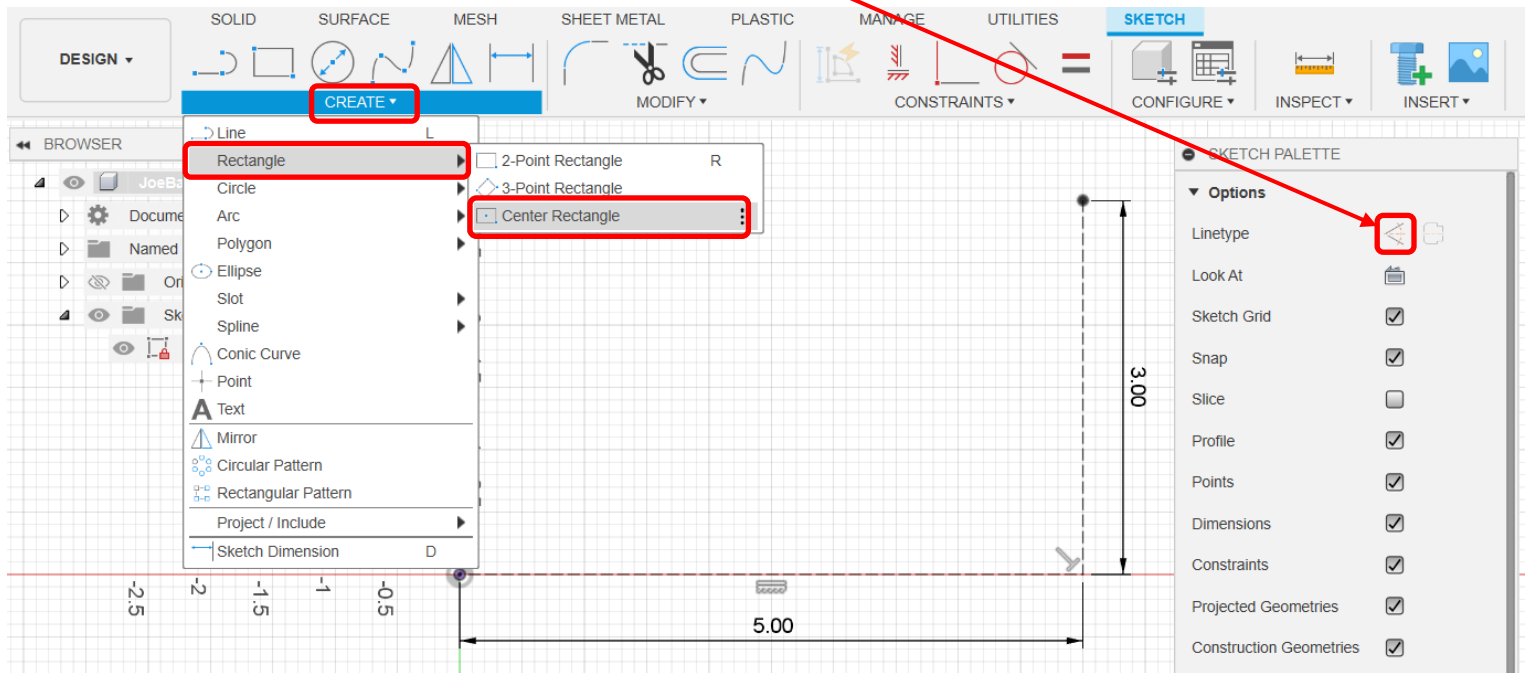
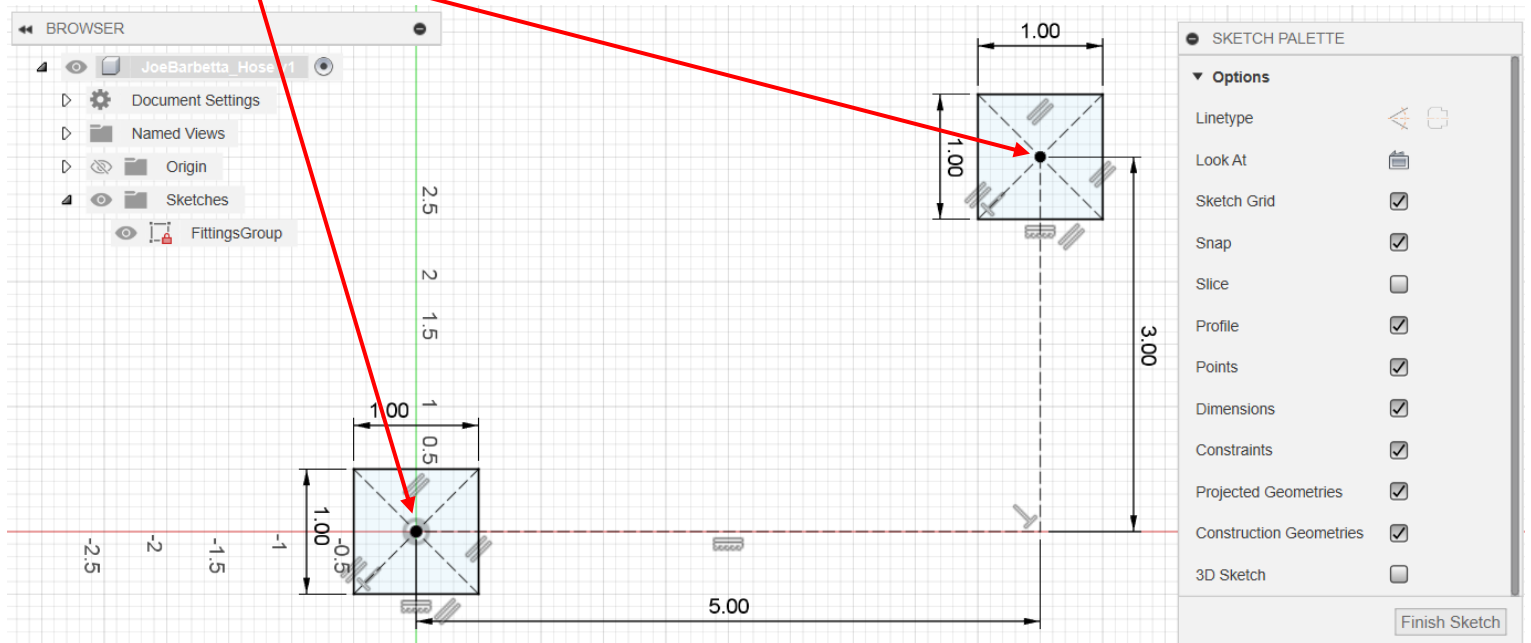Note that the lines are dashed because Construction lines are being drawn.

- drag the **5.00 dimension value down** and the **3.00 dimension to the right** as shown below. This is done just to clean up the sketch.
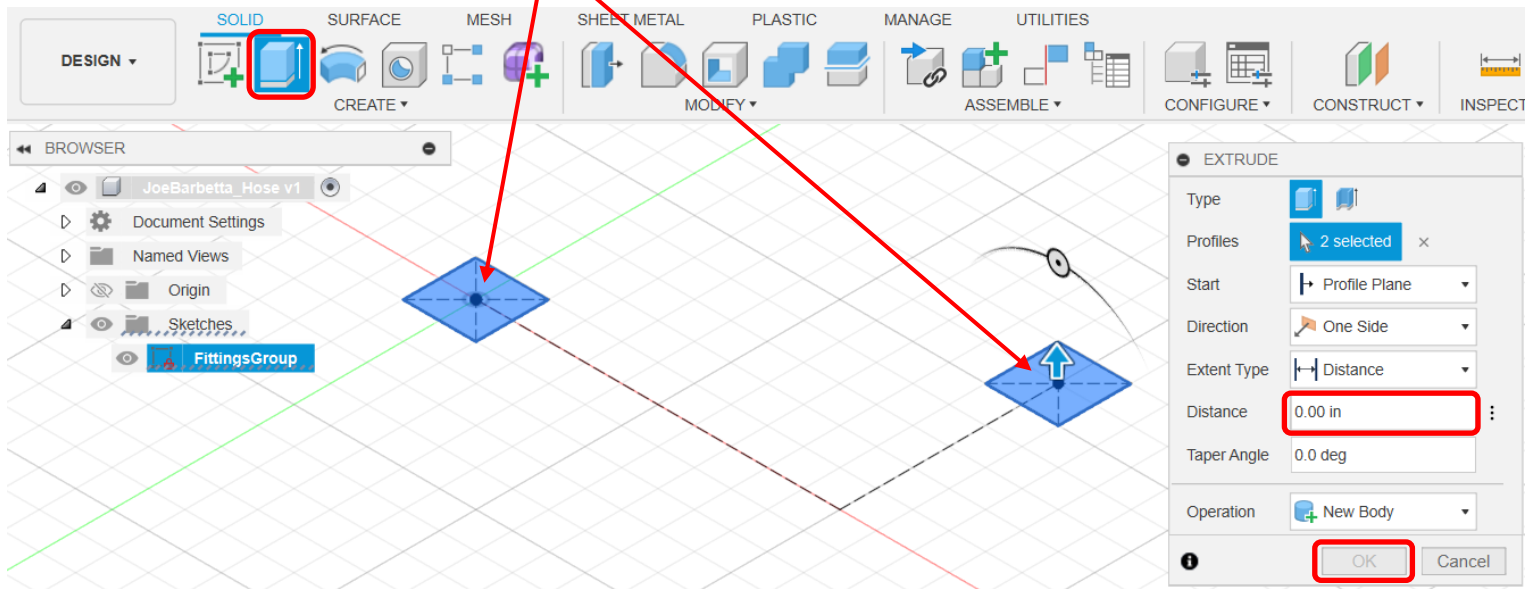
- click on the **Construction** icon again to remove the highlighting. Lines drawn now should be solid.
- from the **CREATE** menu select **Rectangle** and **Center Rectangle**

- click on the **Origin** to define the center point of the rectangle
- extend the rectangle out, type **1**, press the **Tab key**, type **1** again, and press the **Enter key**. The Tab key allows one to toggle between the height and width of the rectangle.
- at the **end of the vertical line** perform the same operation to create a 1 x 1 rectangle there as well

- select the **Extrude** tool and click on **each rectangle** to cause them to turn blue
- in the **Distance** box enter **1** and click **OK**. There should now be 2 gray cubes that are 1 in x 1 in.
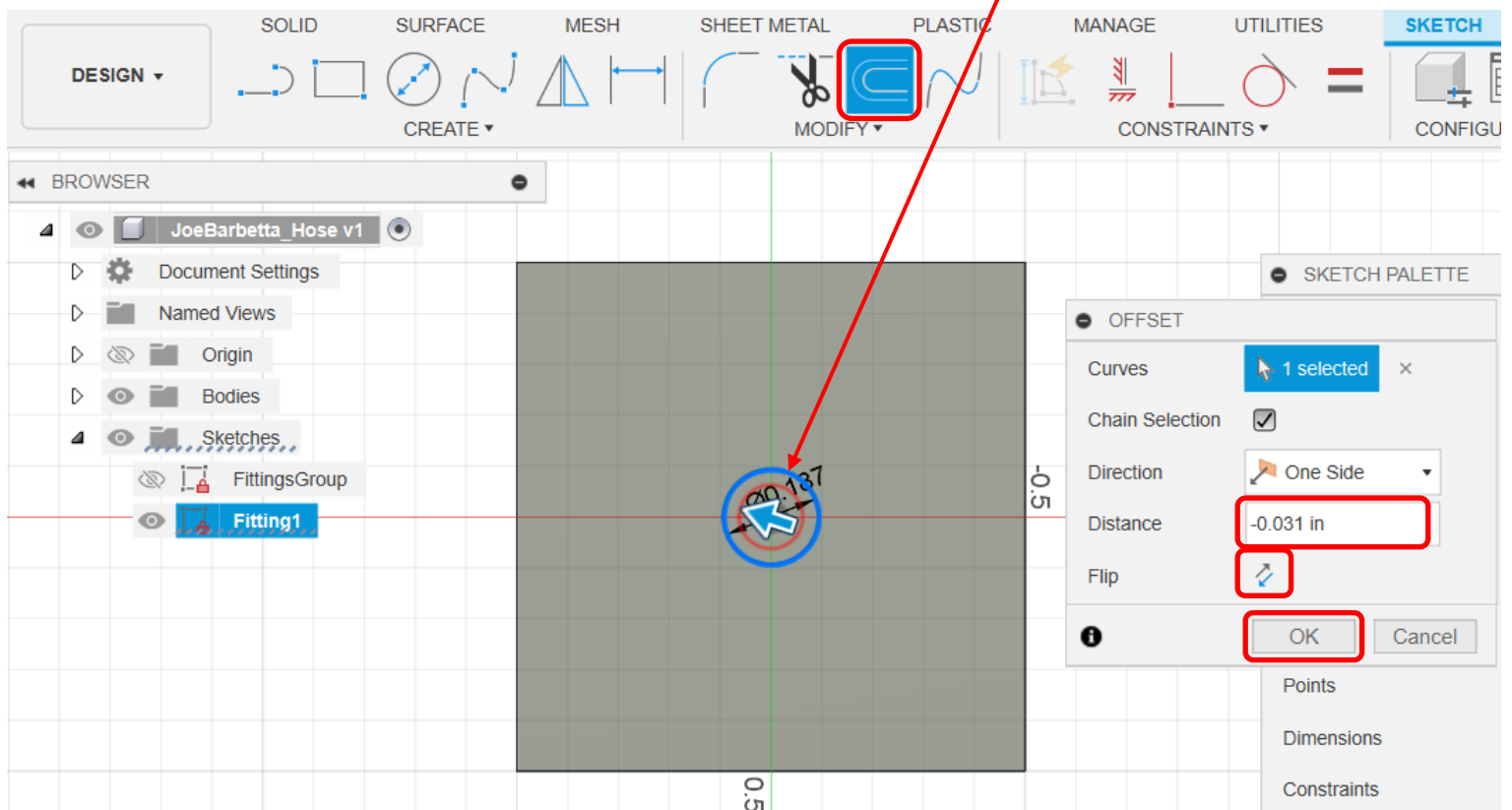
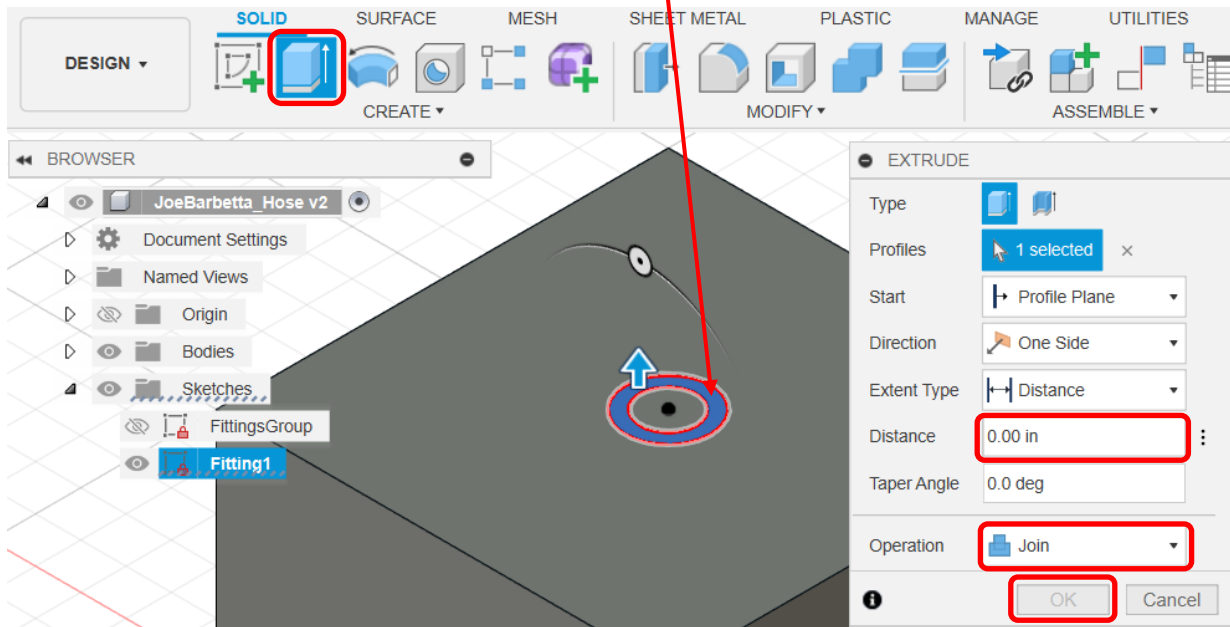- select **Create Sketch** and click on the **top face** of the left cube.

- select the **Center Diameter Circle**. If it is not visible, find it in the CREATE menu.
- click on the **center of the cube surface**, which is the origin in this case, and extend the circle outward
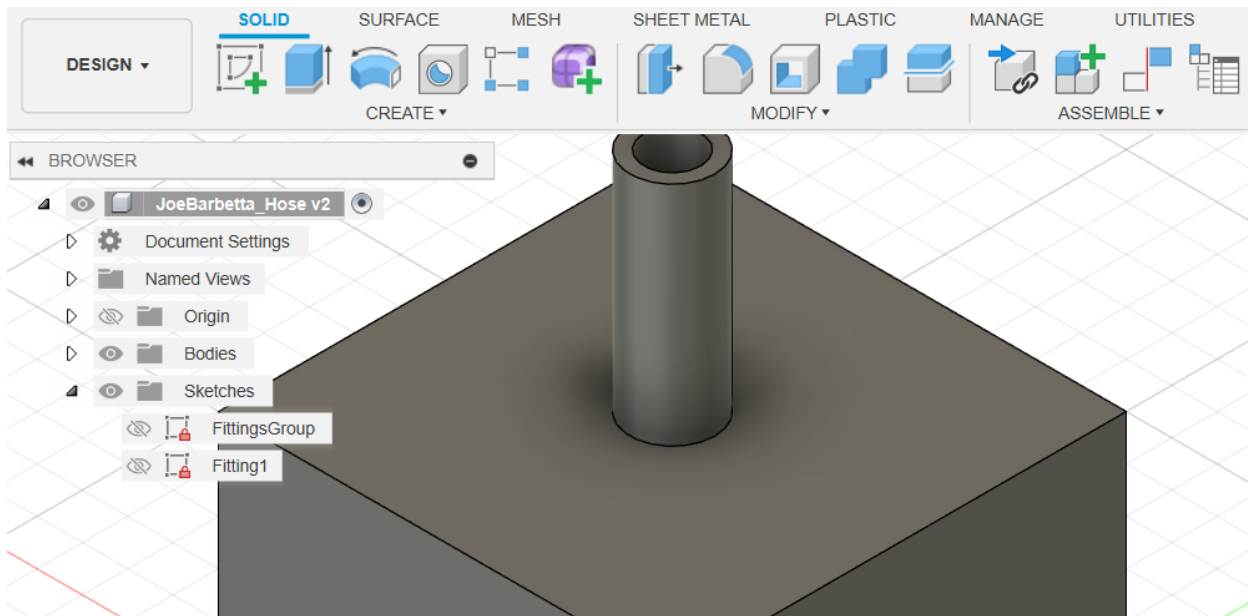- enter **0.187**

- select the **Offset** tool. If it is not visible, find it in the **MODIFY** menu, and click on the **circle** just created
- enter a value of **-0.031** (note the minus sign), for **Distance** which should result in a red circle to appear inside of the blue circle. If needed, one can click on the Flip icon to move the red circle inside.
- click **OK**

- click on the **Home** icon at the **View Cube**
- select the **Extrude** tool and click on the **region between the two circles**
- enter **0.5** for **Distance**
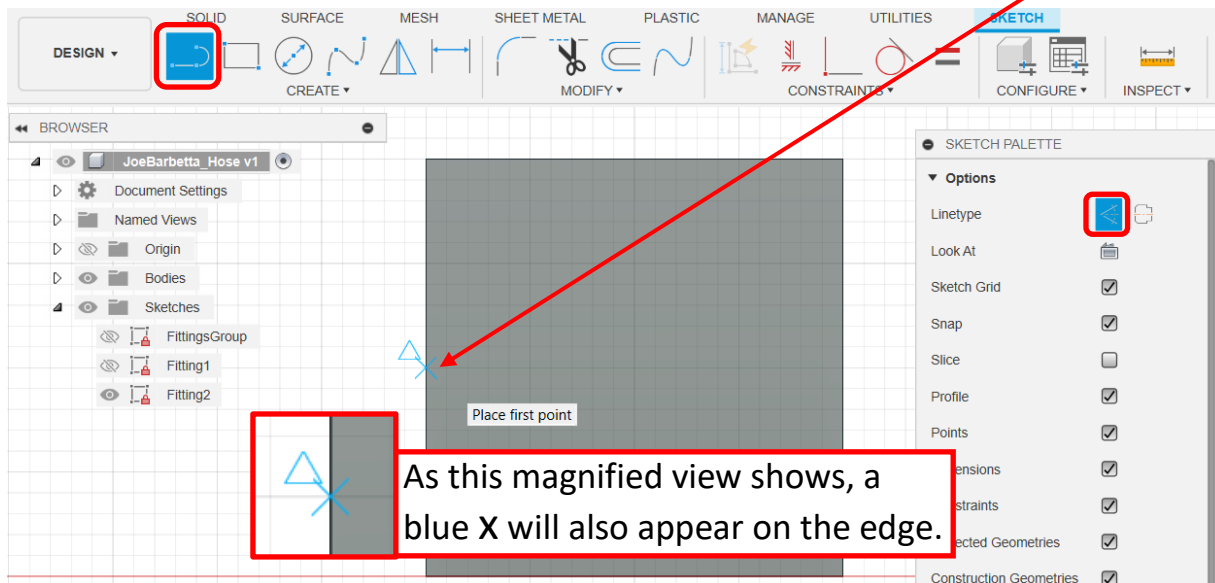- ensure the **Operation** is **Join** and click **OK**
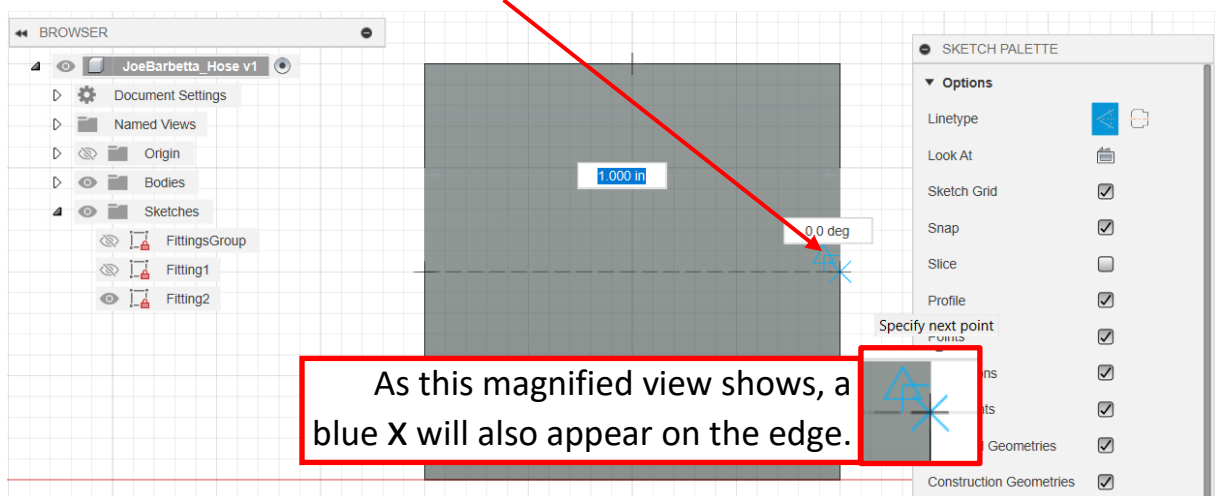

The result should look like that below.

- zoom into the 2nd cube
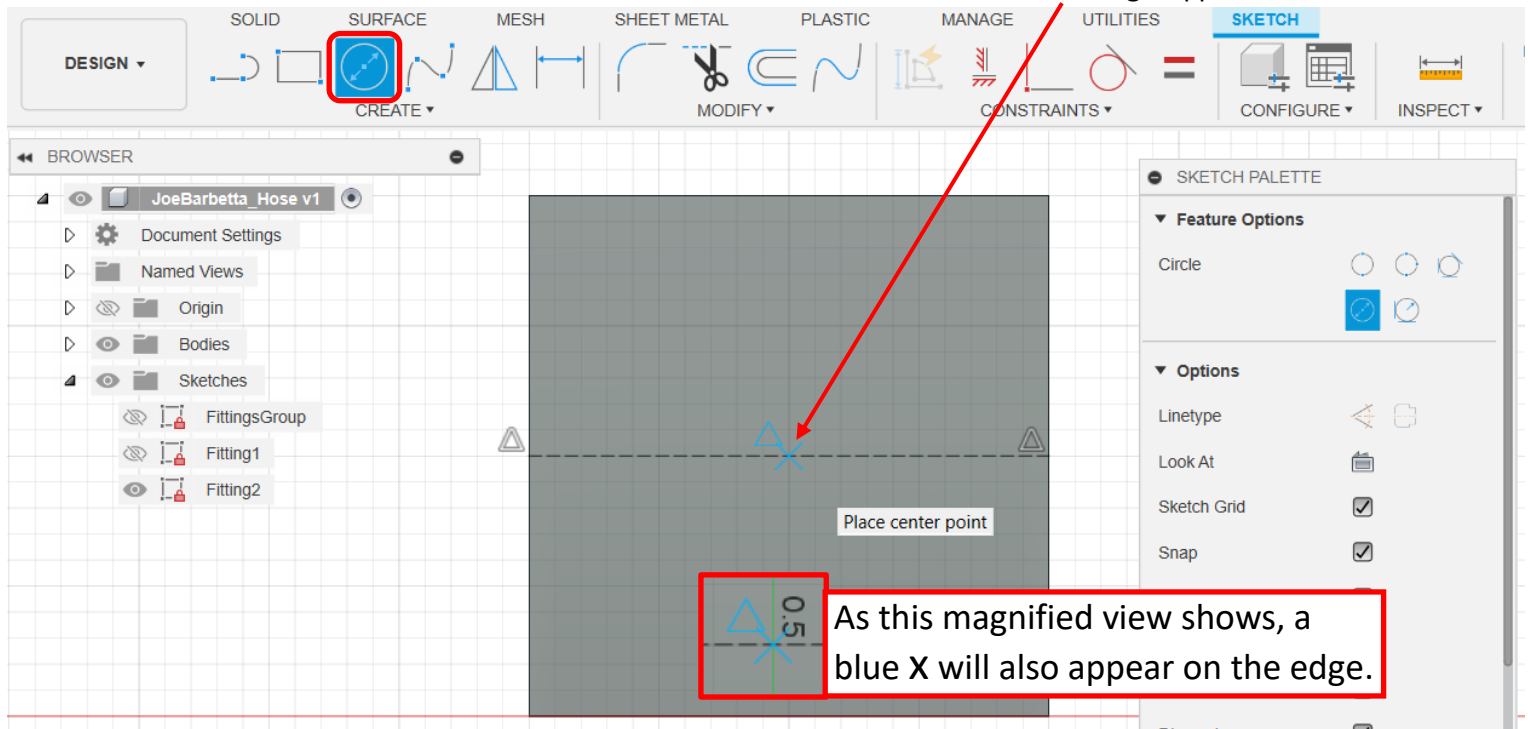- select **Create Sketch** and click on the **side face** of the cube

- click on the **Construction** icon to highlight it blue
- select the **Line** tool and move the mouse along the left edge of the cube until the **blue triangle** shows and click on that spot
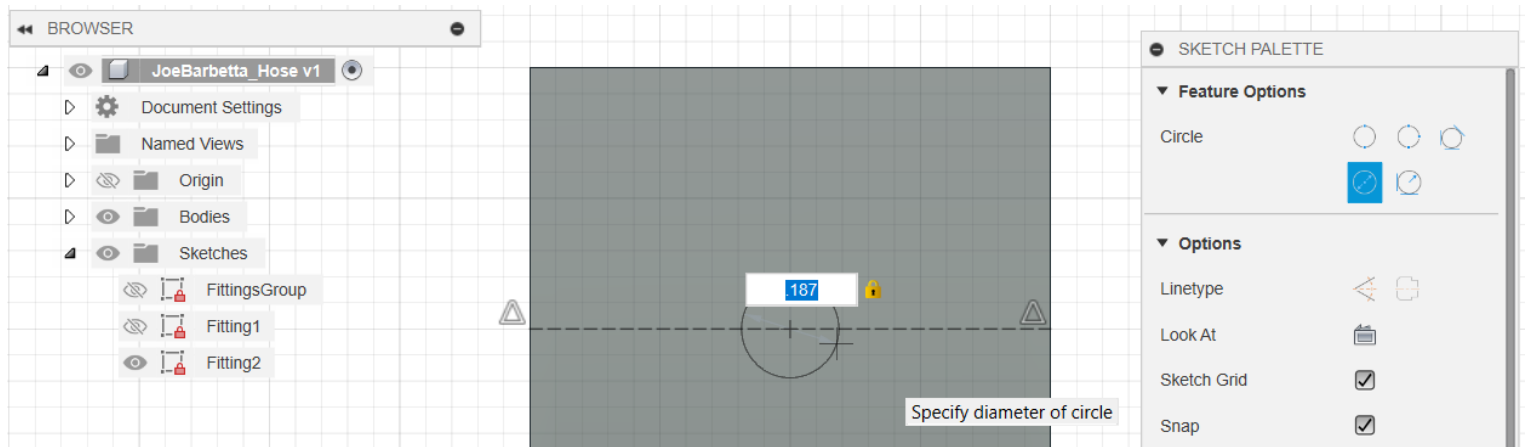

As this magnified view shows, a blue **X** will also appear on the edge.

- extend the line **to the right** until a **blue triangle** appears and then **click on that spot**

As this magnified view shows, a blue **X** will also appear on the edge.

- click on the **Construction** icon again to **turn off the blue highlighting**
- select the **Center Diameter Circle**. Move the mouse over the dashed line until the **blue triangle** appears and then click there

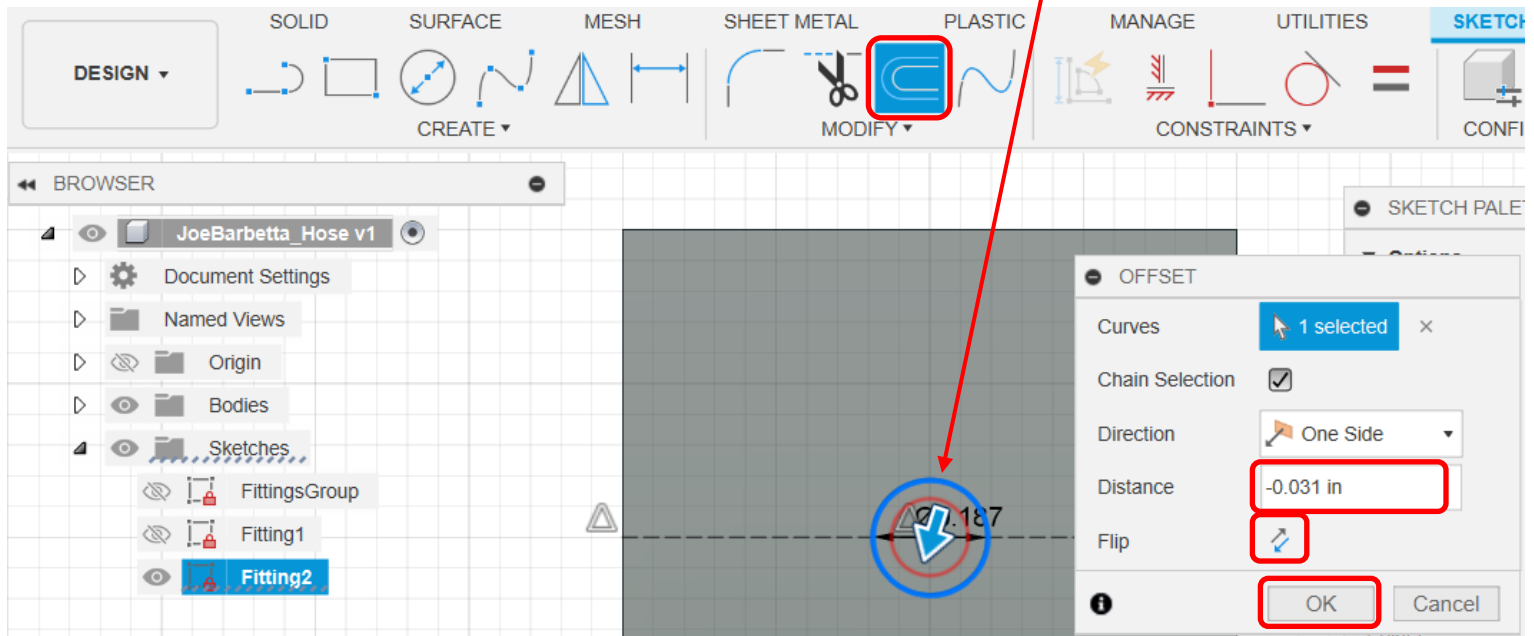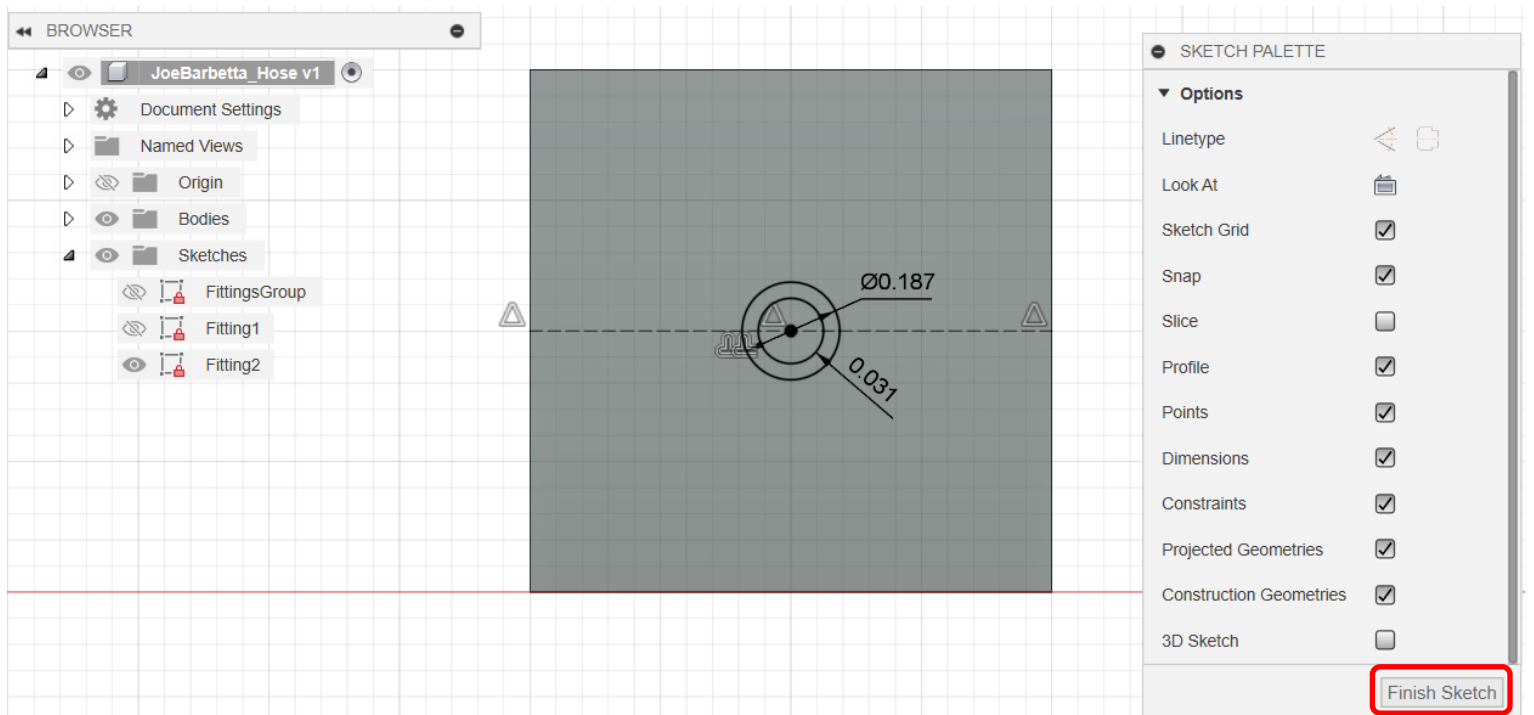As this magnified view shows, a blue **X** will also appear on the edge.

- extend the circle outward and enter **0.187**

Continued on the next page.

- select the **Offset** tool. If it is not visible, find it in the **MODIFY** menu, and click on the **circle** just created
- enter a value of **-0.031** (note the minus sign), for **Distance** which should result in a red circle to appear inside of the blue circle. If needed, one can click on the Flip icon to move the red circle inside.
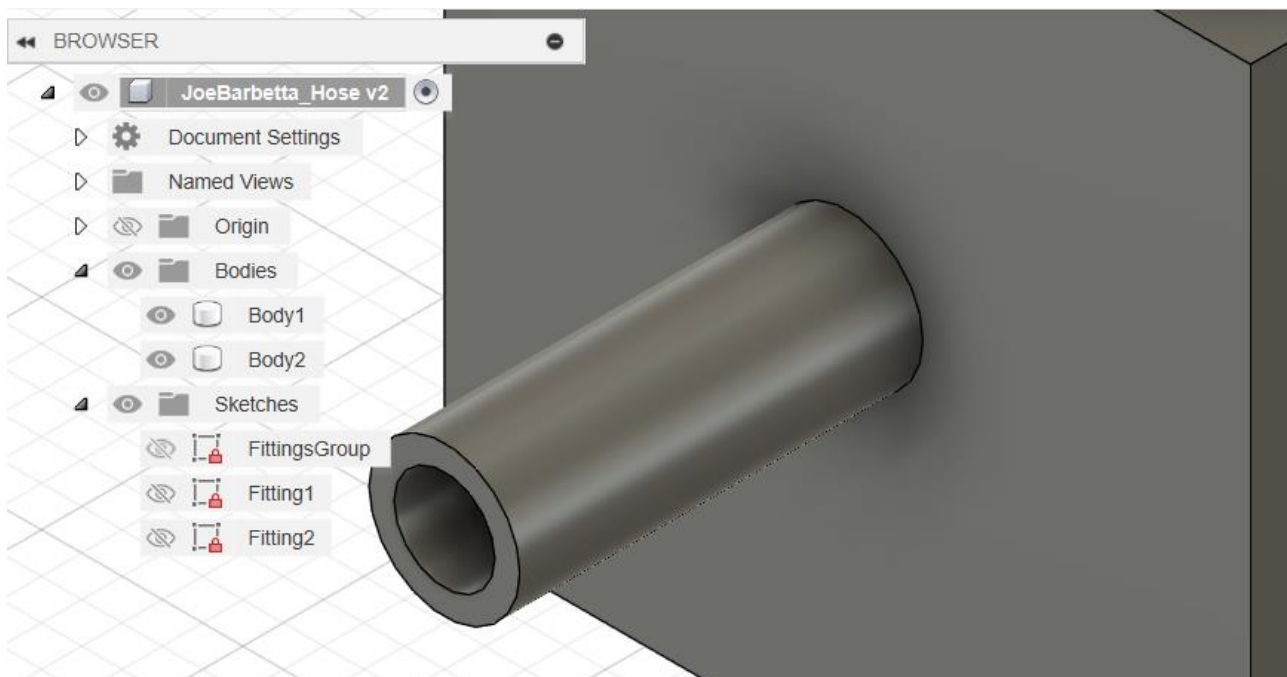- click **OK**

- drag the **2 dimension values** to clean up the Sketch and click **Finish Sketch**

- click on the **Home** icon at the **View Cube**
- select the **Extrude** tool and click on the **region between the two circles**
- enter **0.5** for **Distance**
- ensure the **Operation** is **Join** and click **OK**

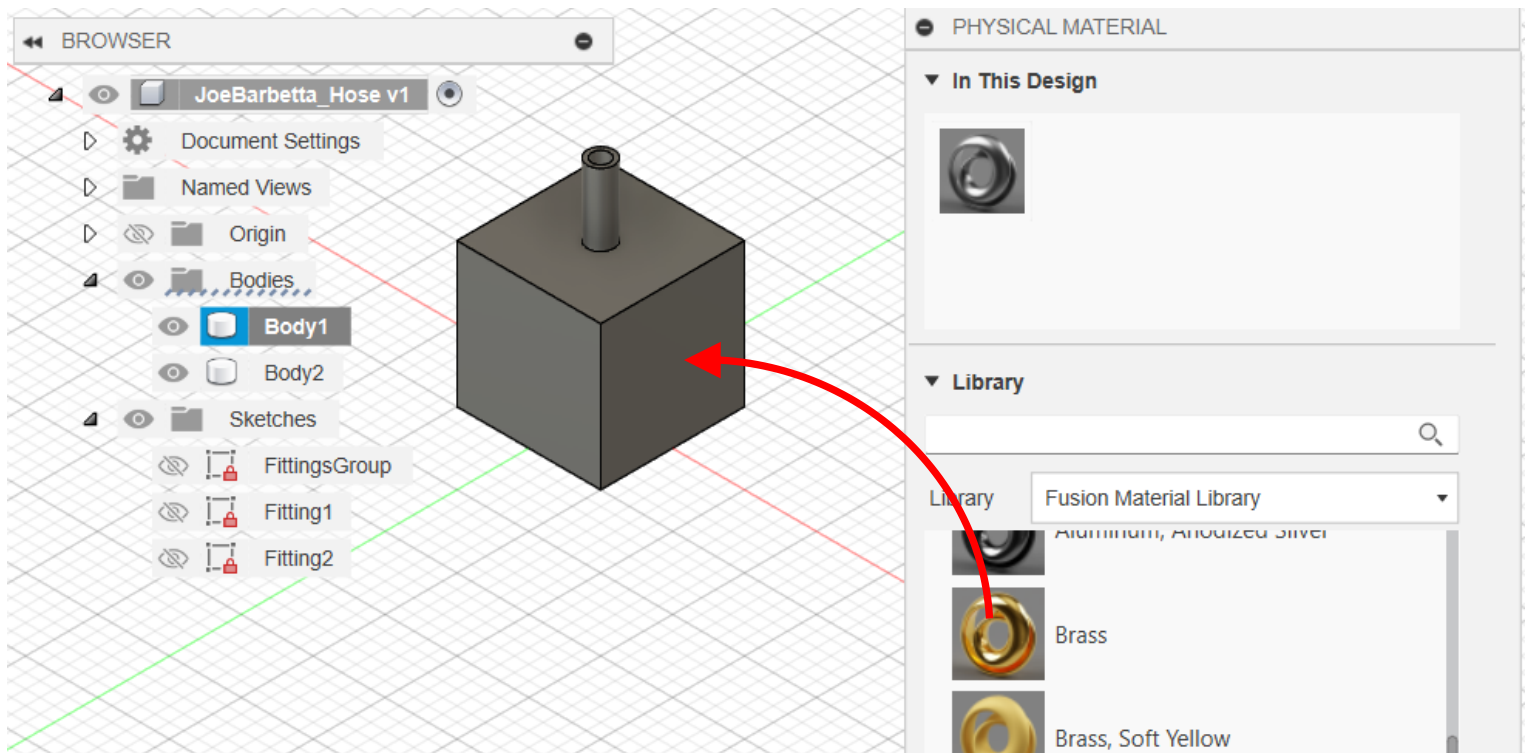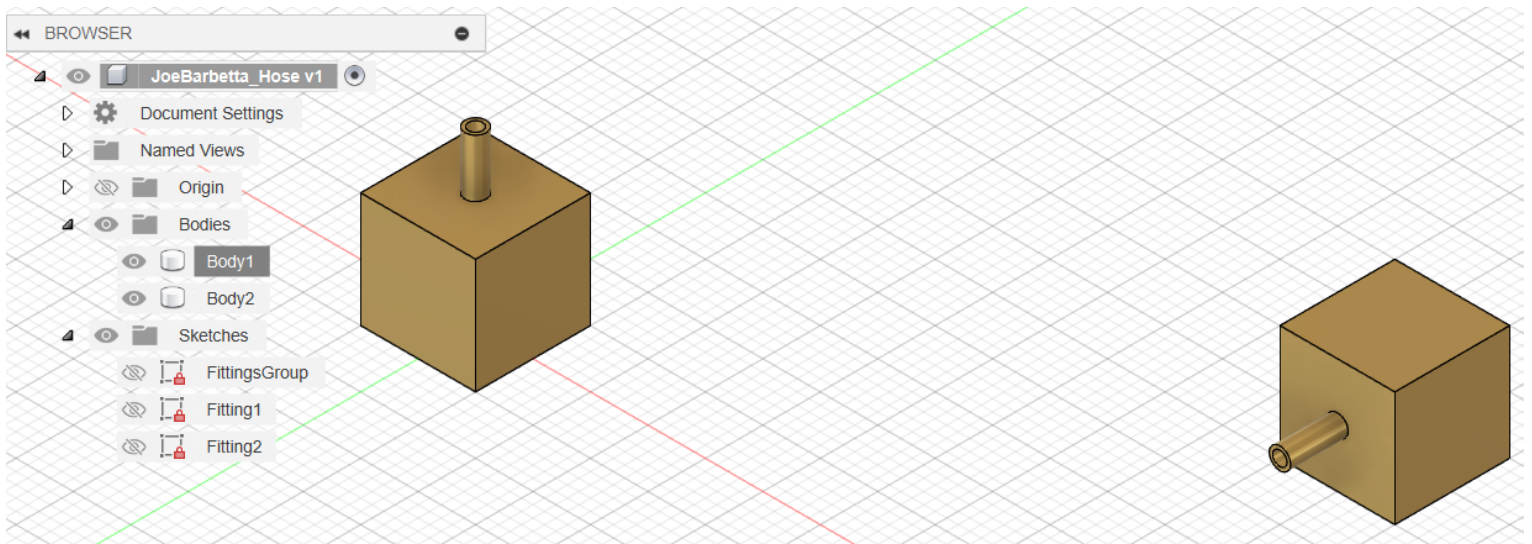This should be the result of the Extrude operation.

Setting the Material

- click on the **Home** icon at the **View Cube**
- click on the **arrow** for **Bodies** to open that group
- right-click on **Body1** and select **Physical Material**

- scroll down to the **Metal** folder and click on it to access the metal selections
- scroll down again to find **Brass**
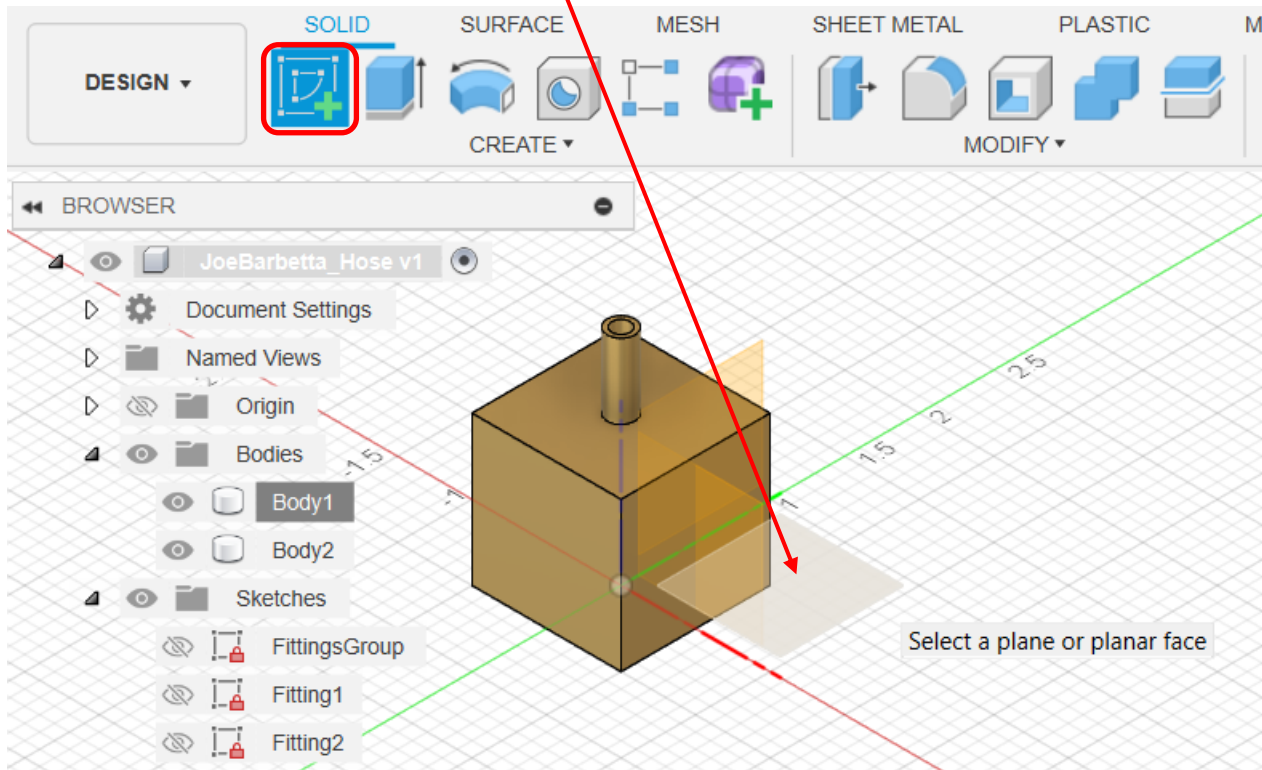- **drag its icon onto the cube**

- drag the **Brass icon onto the other cube** as well and close the PHYSICAL MATERIAL window
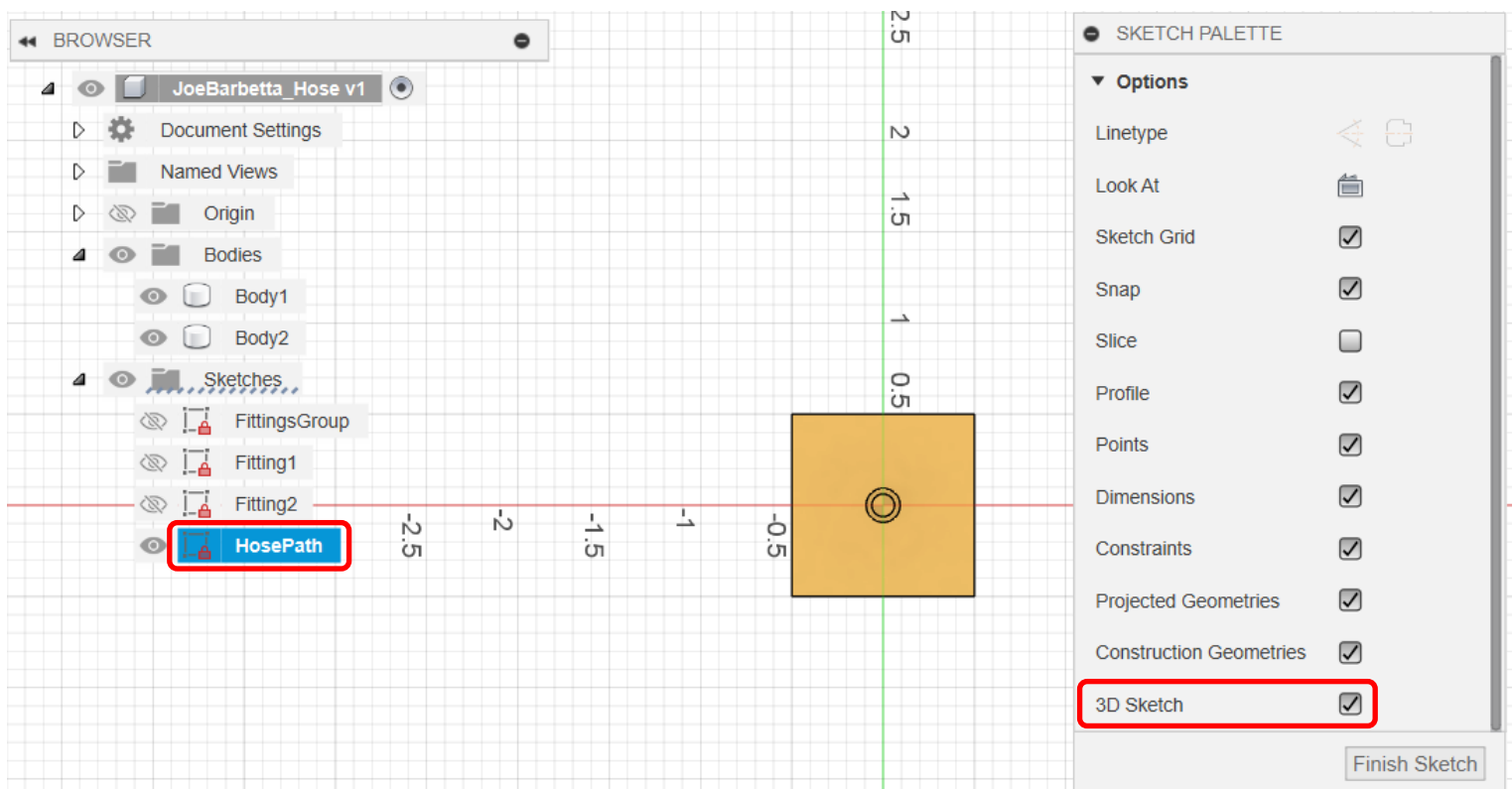The result appears below.
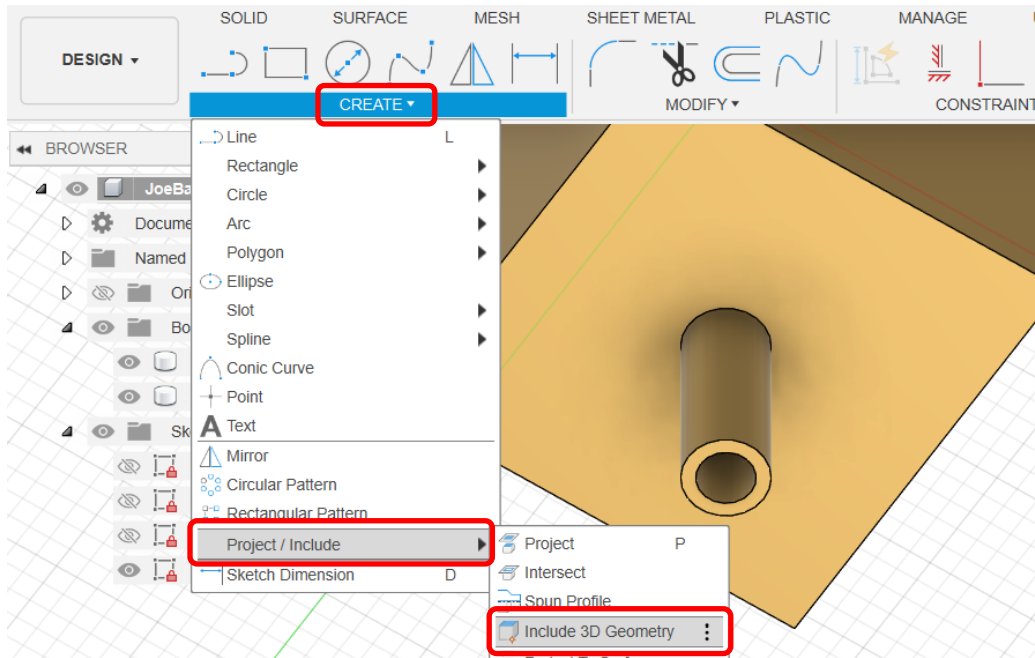
Creating a 3D Sketch for the Tube Path

- zoom into the **left cube** as shown below.
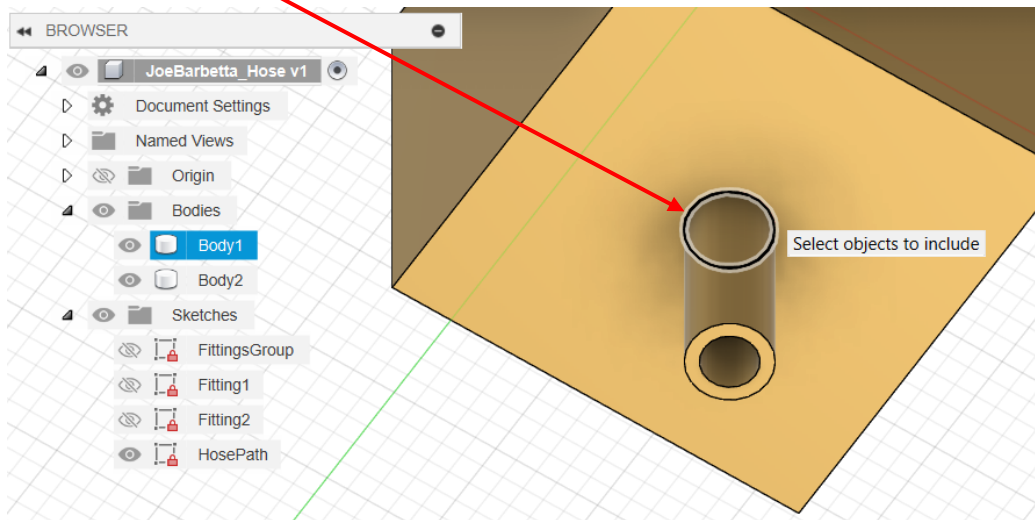- select **Create Sketch** and click on the **bottom rhombus**. Do not click on the cube.

- rename the Sketch to **HosePath**
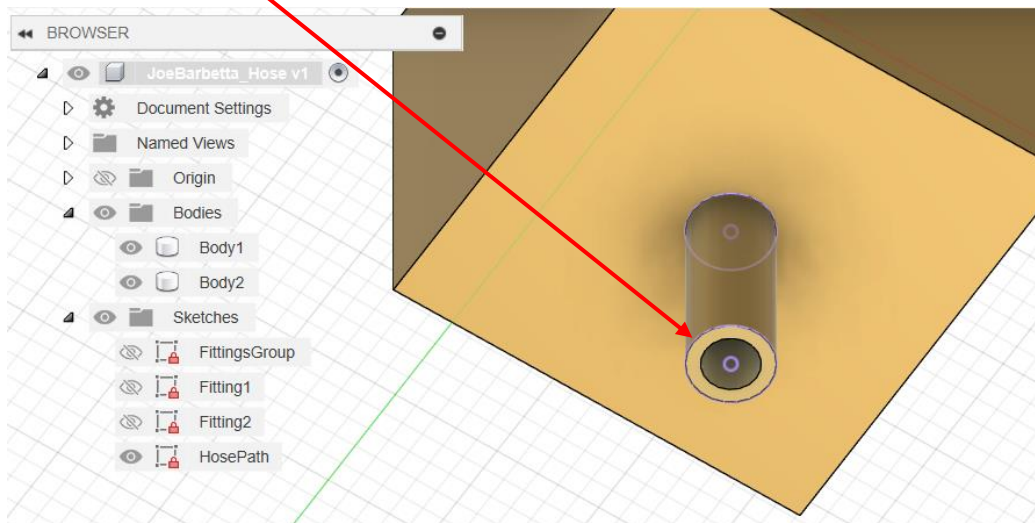- check the **3D Sketch** checkbox

- turn the **View Cube** to achieve a view similar to that below
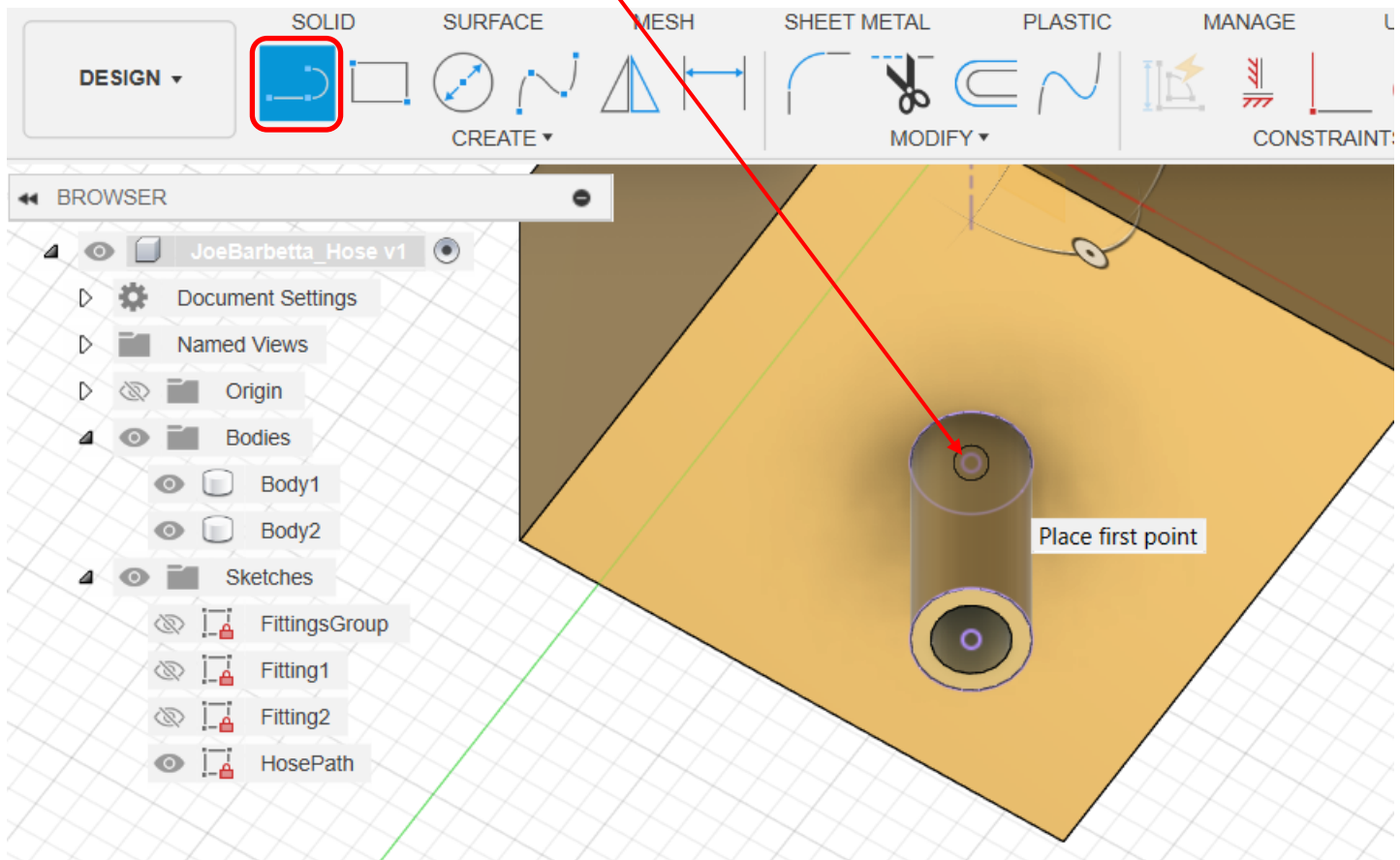- from the **CREATE** menu select **Project / Include** and Include **3D Geometry**

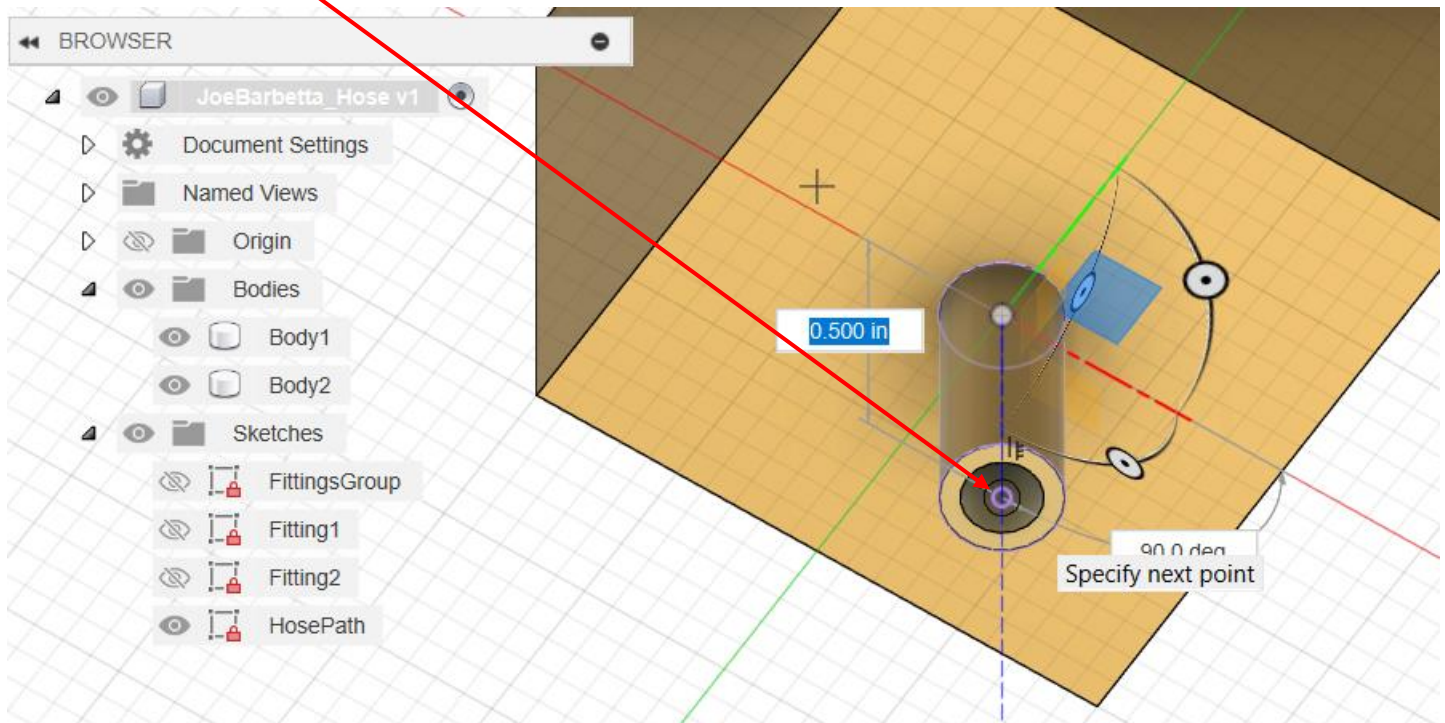- click on the **intersection of the extruded tube with the cube**

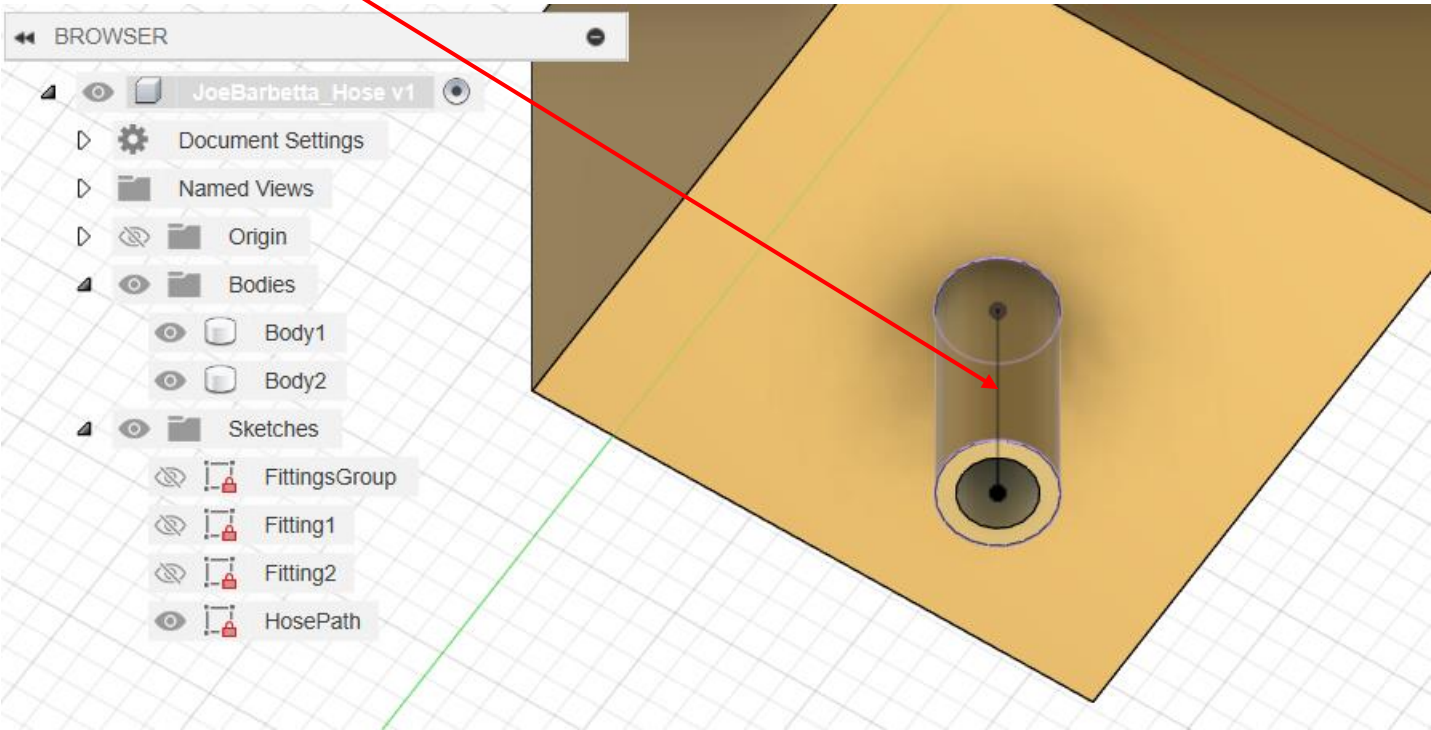- click on the **outer edge at the end of the extruded tube**

- select the **Line** tool and click on the point in the **center of the circle** indicated with the arrow
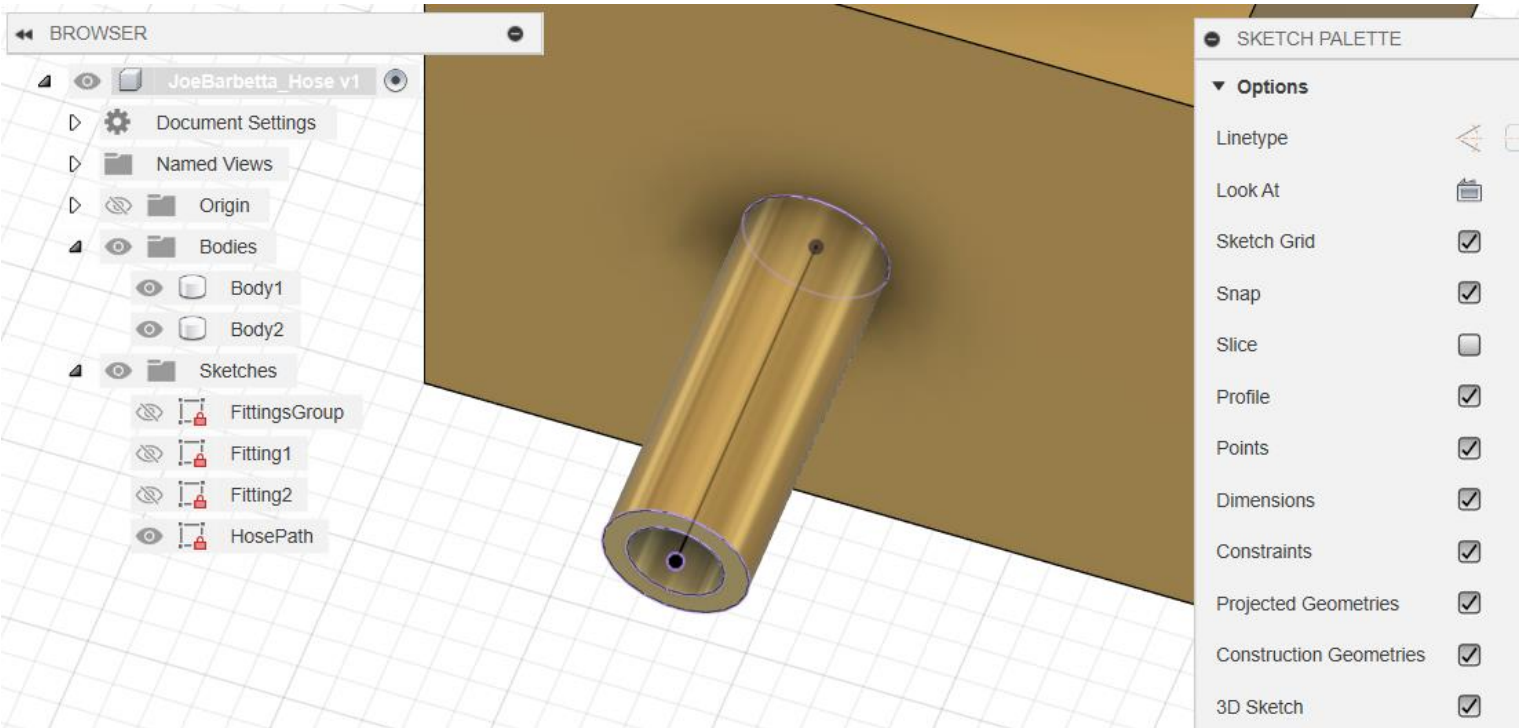

- click on **center of the circle** indicated by the arrow and press the **Esc Key**. The value of 0.500 does not need to be entered. It will show automatically.
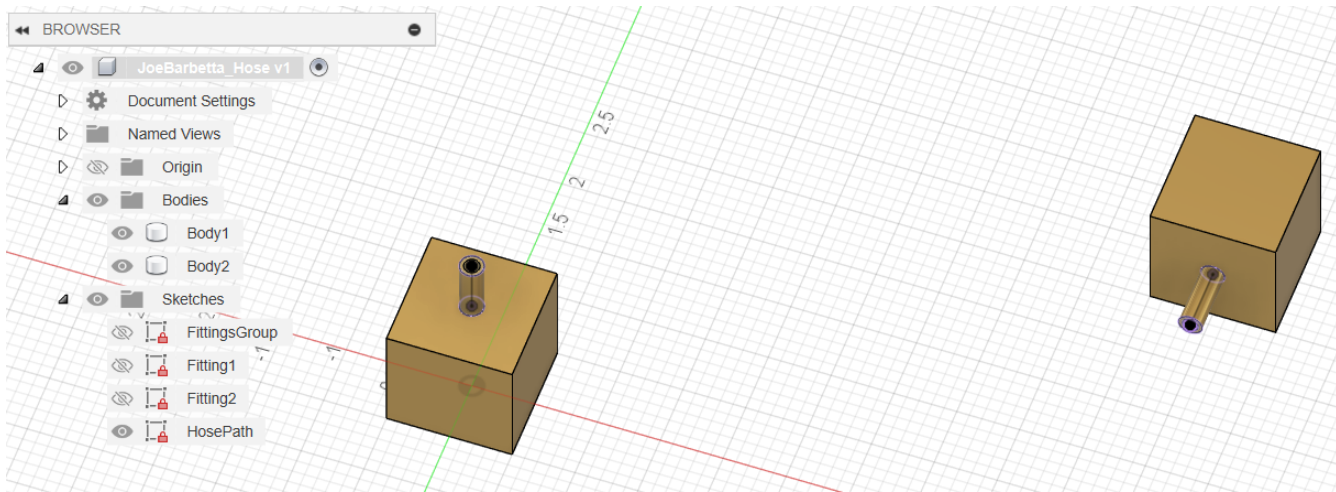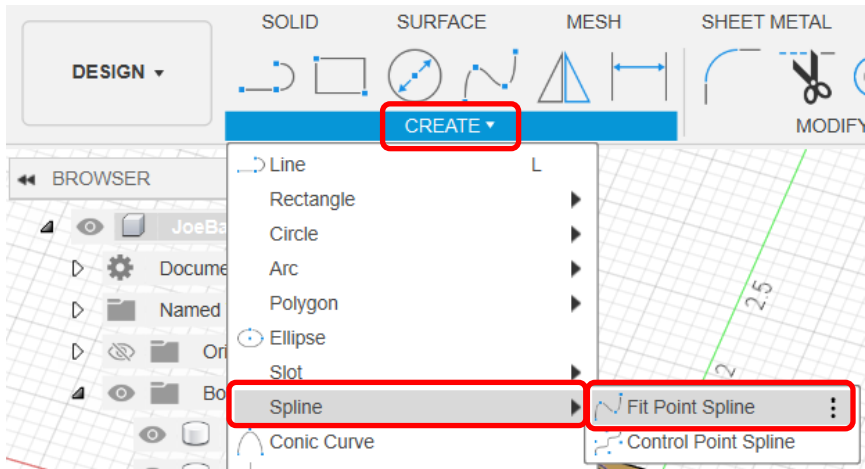
This is the result of the line created.

- zoom into the extruded tube of the other body and perform the same operation

- zoom out to view both cubes. **Note that Finish Sketch should Not be clicked yet.**
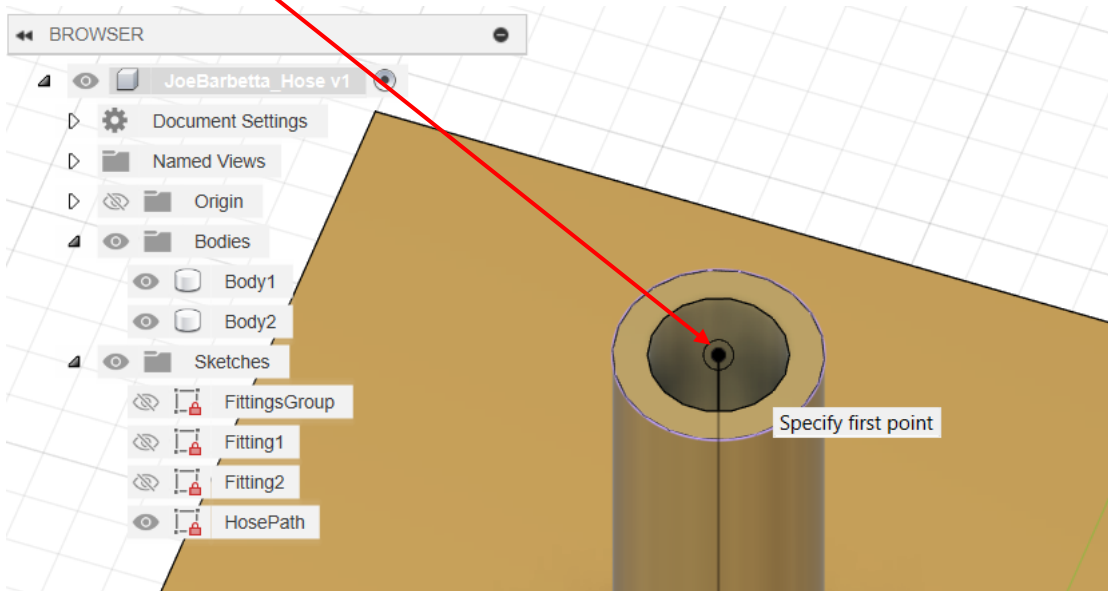

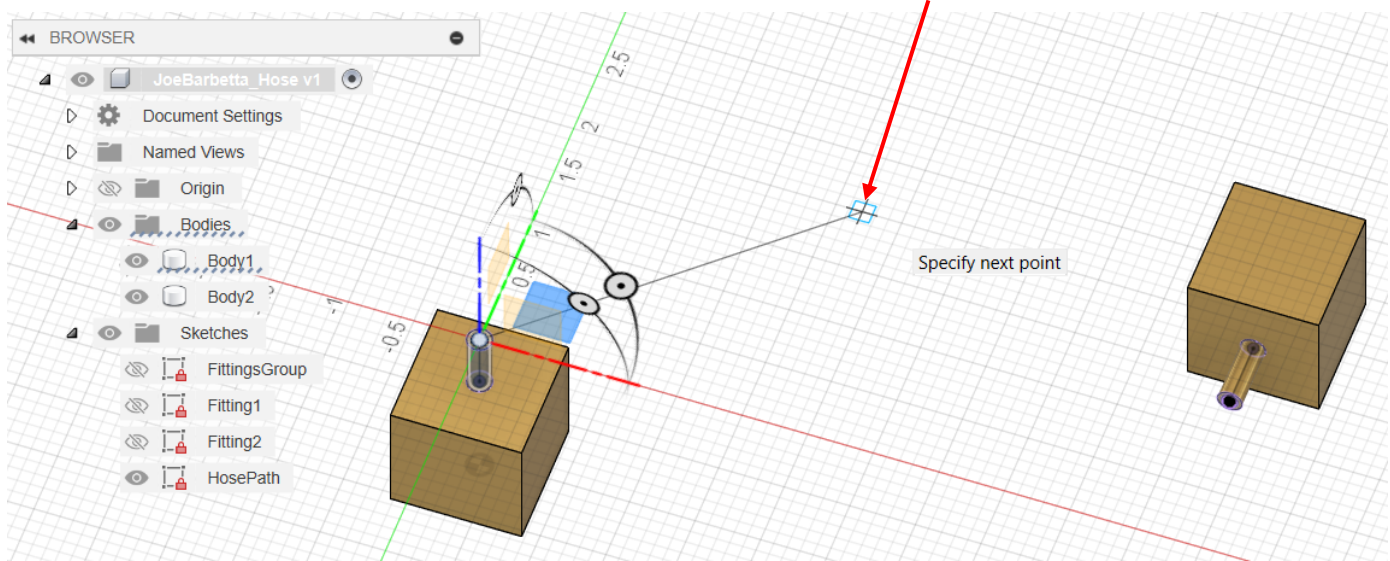- from the **CREATE** menu select **Spline** and **Fit Point Spline**

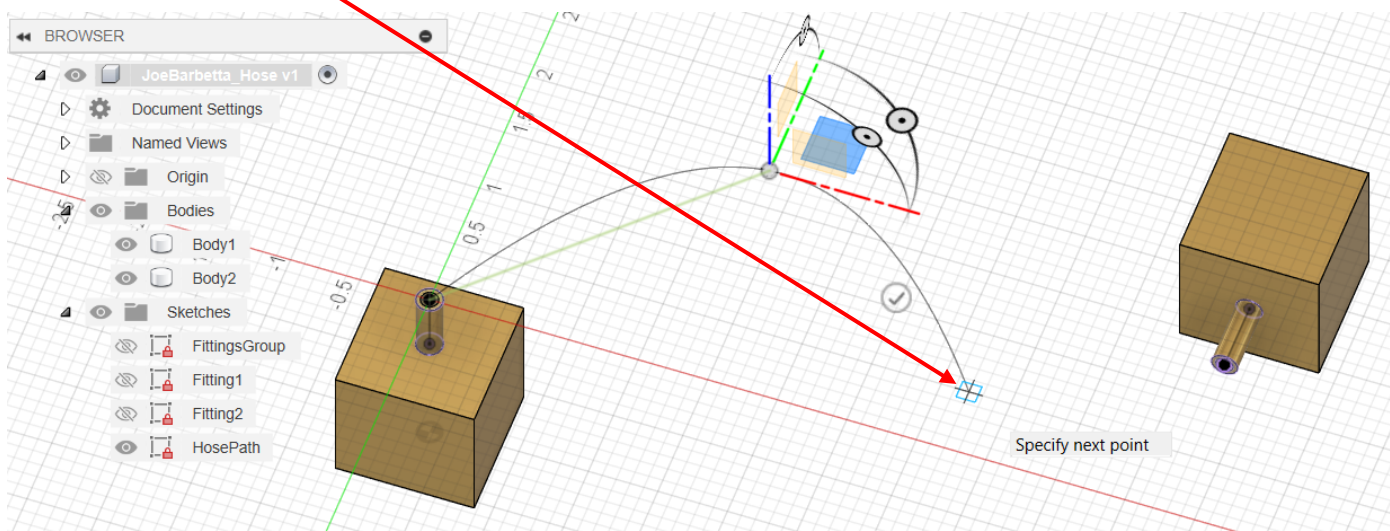- zoom into the extruded tube on the top of the left cube
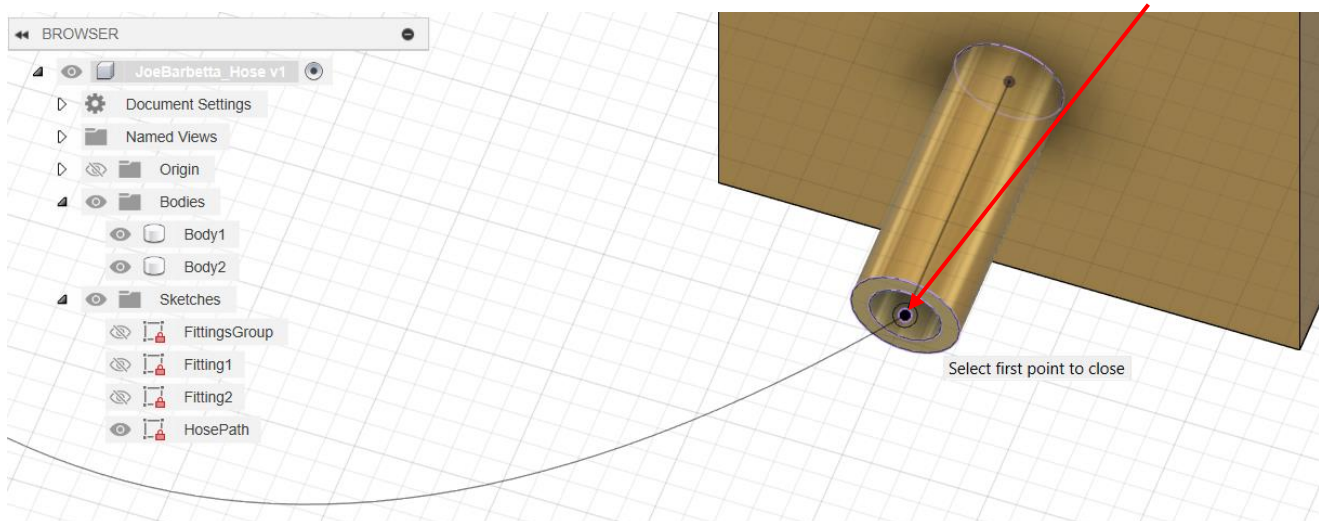- click on the **top endpoint** of the line

- zoom out using the Mousewheel to view both view cubes and **click in the area shown**. The position is not critical.
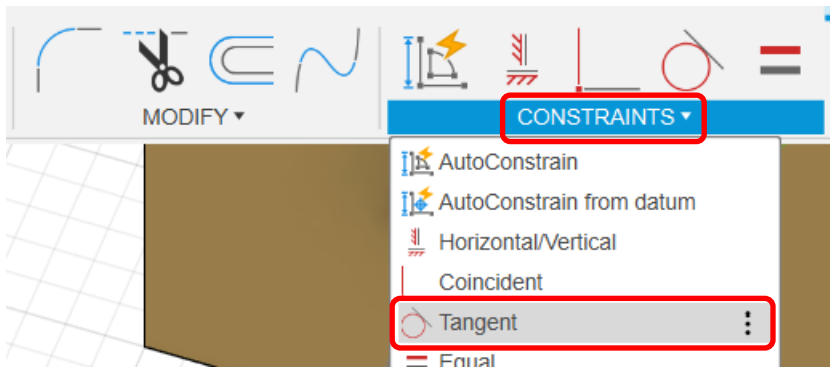

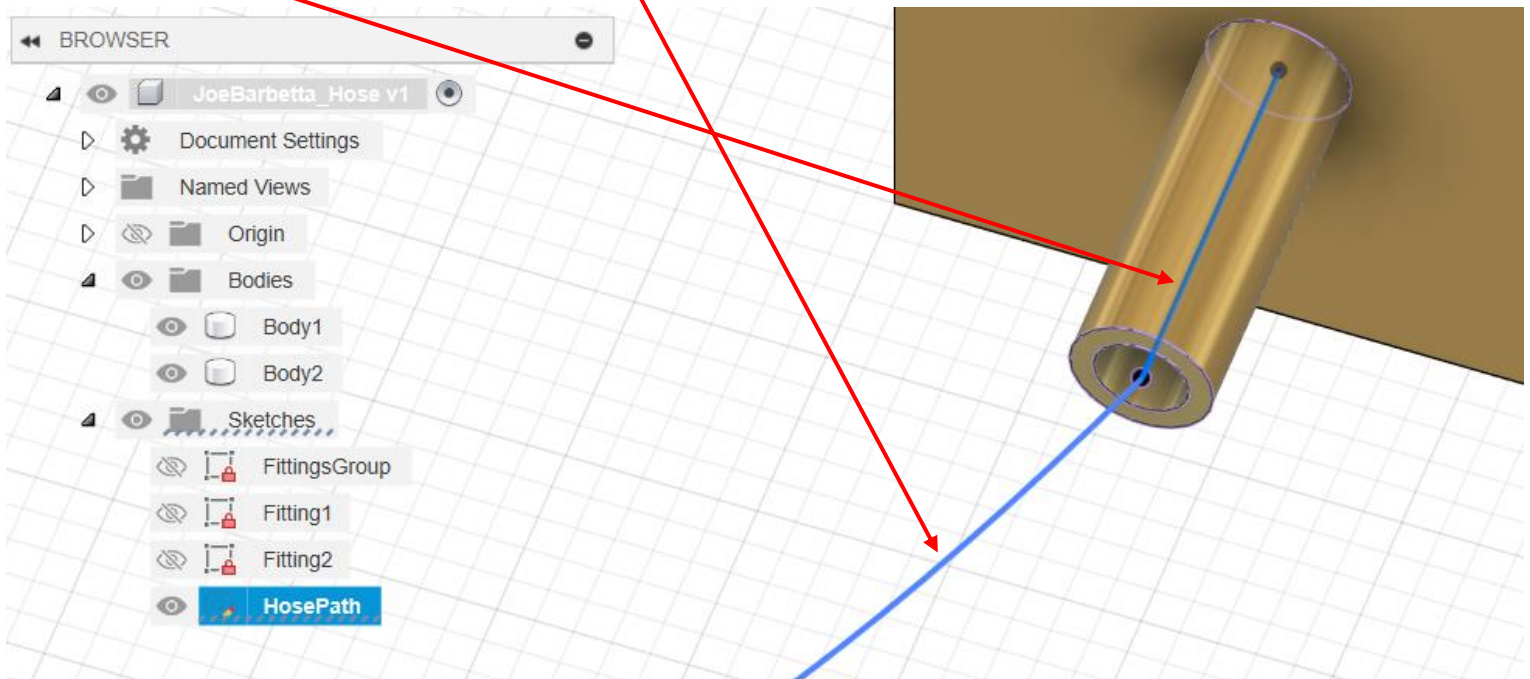- **click again in the area shown**. Again, the position is not critical.

- using the Mousewheel zoom into the the extruded tube on the right cube and **click on the endpoint** of its line
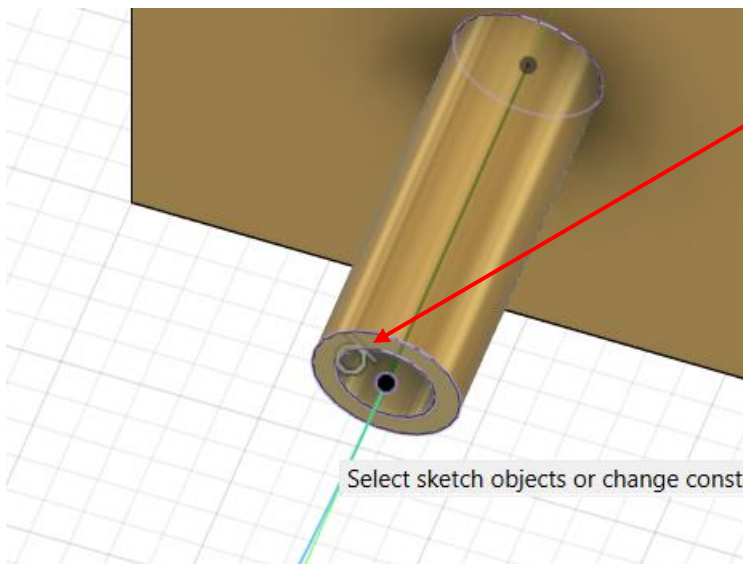
- from the **CONSTRAINTS** menu select **Tangent**

- click on the **straight line** and then click on the **spline curve**

The part of the curve should straighten in-line with the tube line and the **Tangent symbol** should appear.
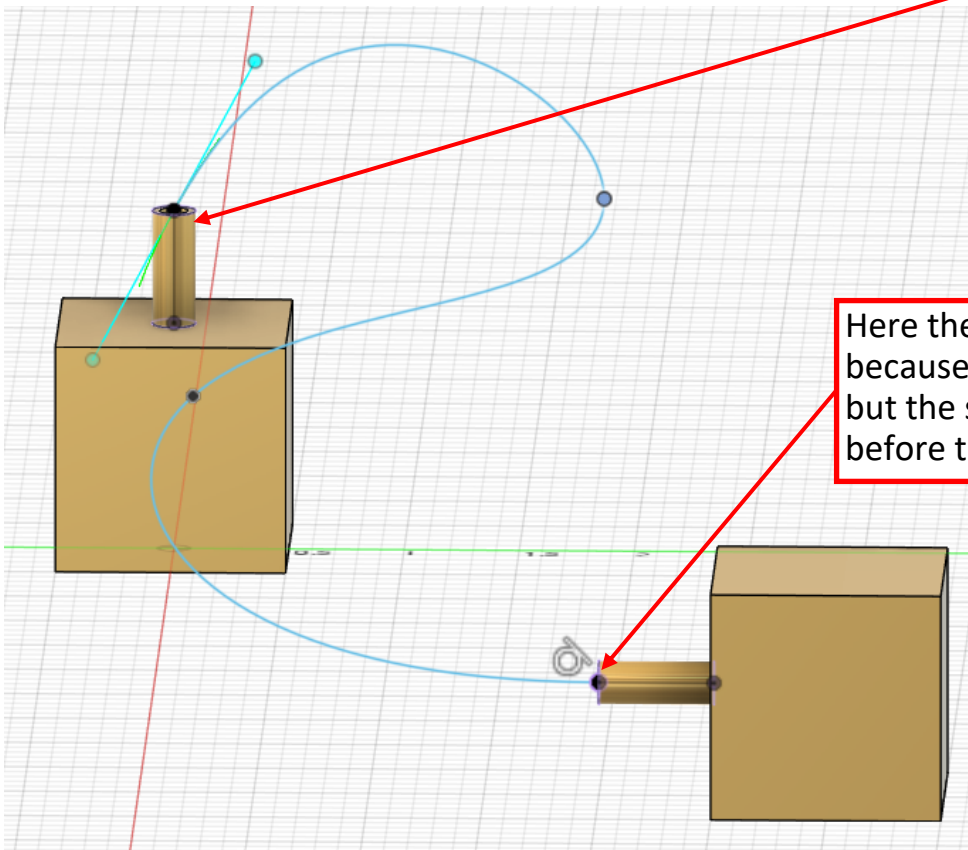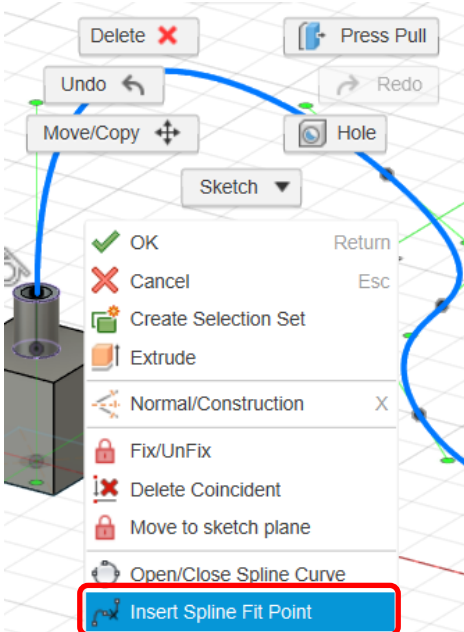
These next steps will be annoying.

The spline was created as a 3D Sketch, which is not as intuitive as working with a 2D Sketch on a plane. However, it can be fun manipulating a spline curve in 3D space. One can change the view between top, front, and left or right while manipulating the curve in 3D space. **Also note that your spline may look different from that shown because the exact points in 3D space for the 2 middle points were not specified.**

The curve has to be adjusted so it is close to being aligned with the top tube of the left cube. The points on the curve can be moved and when a point is selected light blue lines with small blue circles at the ends will appear. These light blue circles can be moved to adjust the curvature at a point. These are called **handles**. After adjusting a handle click on the background.

- as mentioned, adjust the spline so that it is close to being aligned with the **axis of the top tube**
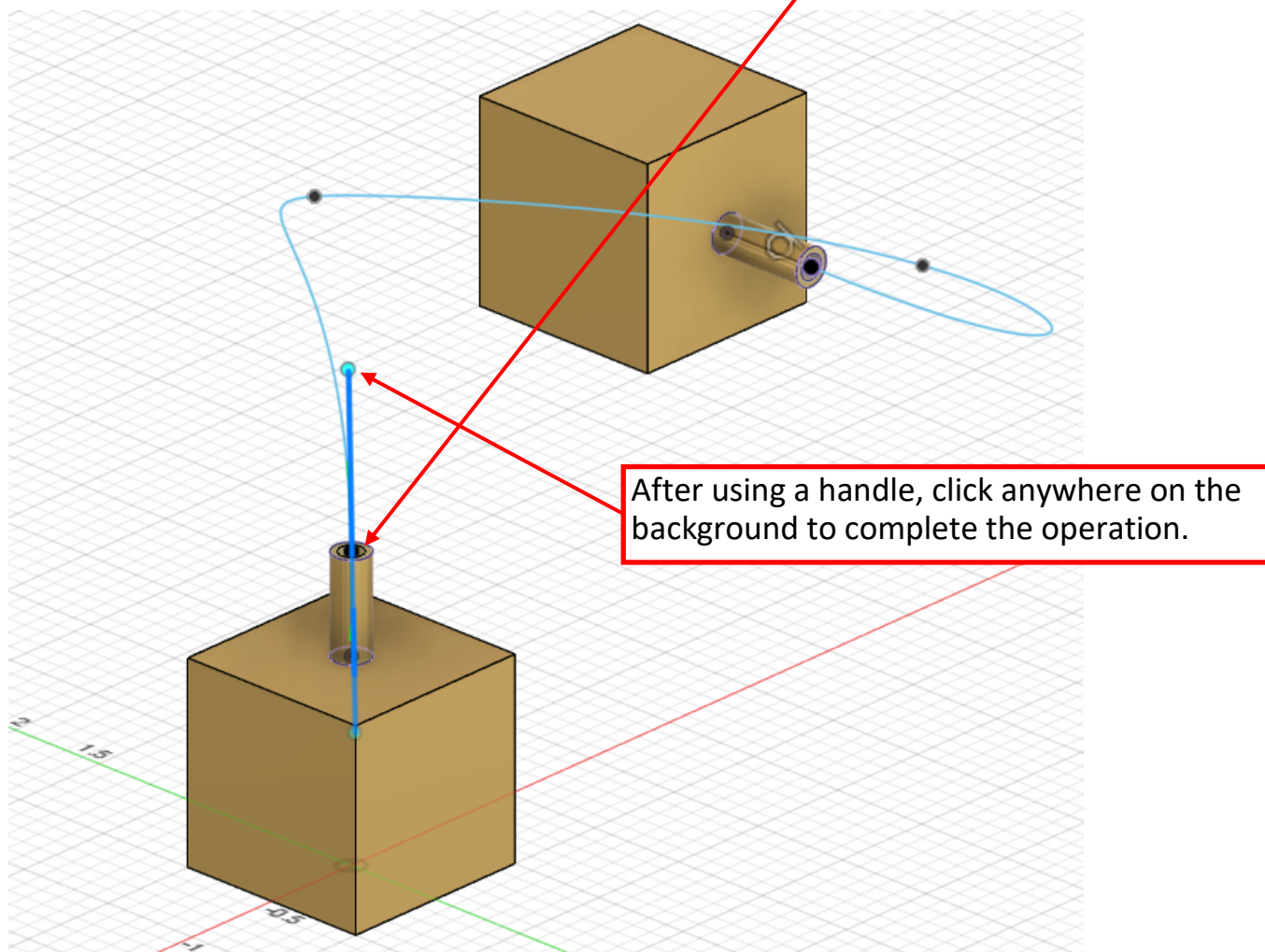

Here the spline is perfectly aligned because a Tangent constraint was added, but the spline has to be somewhat aligned before the Tangent constraint is used.
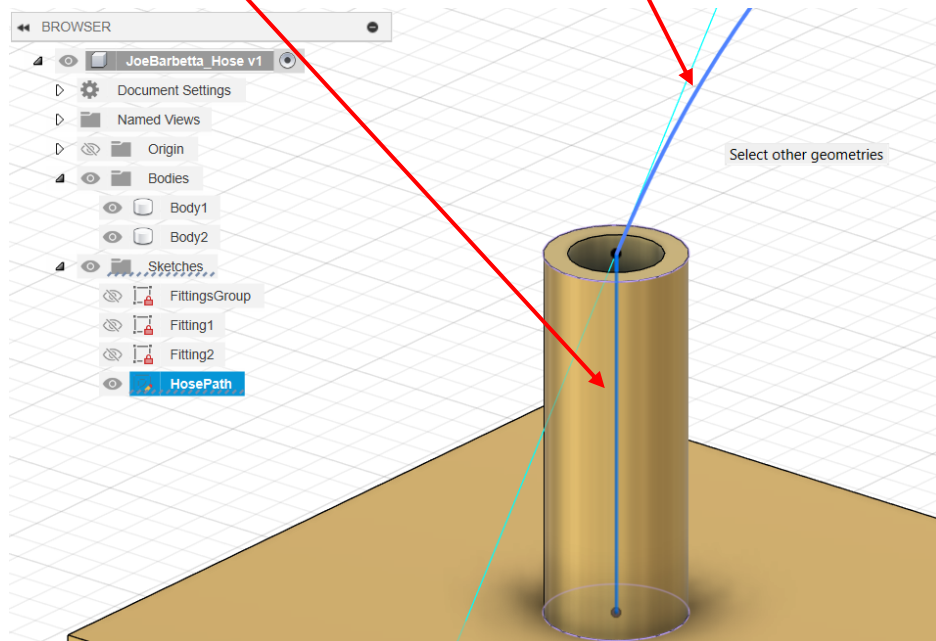

This may not be be needed, but one can right-click on the spline curve and select Insert Spline Fit Point from the pop-up menu. Then one can click anywhere on the spline to add a point. After one or more points are added, press the Esc key to complete the add point operation.
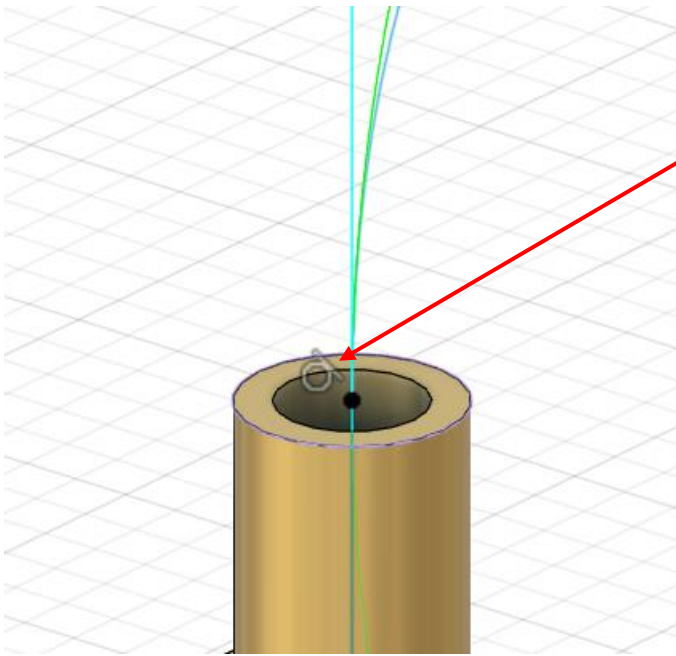
Here is a view where the spline was adjusted so that is is close to being aligned with the tube extrusion. One of the handles was used to help with the adjustment.

After using a handle, click anywhere on the background to complete the operation.

- as done with the 1st extruded tube, select **Tangent** from the **CONSTRAINTS** menu
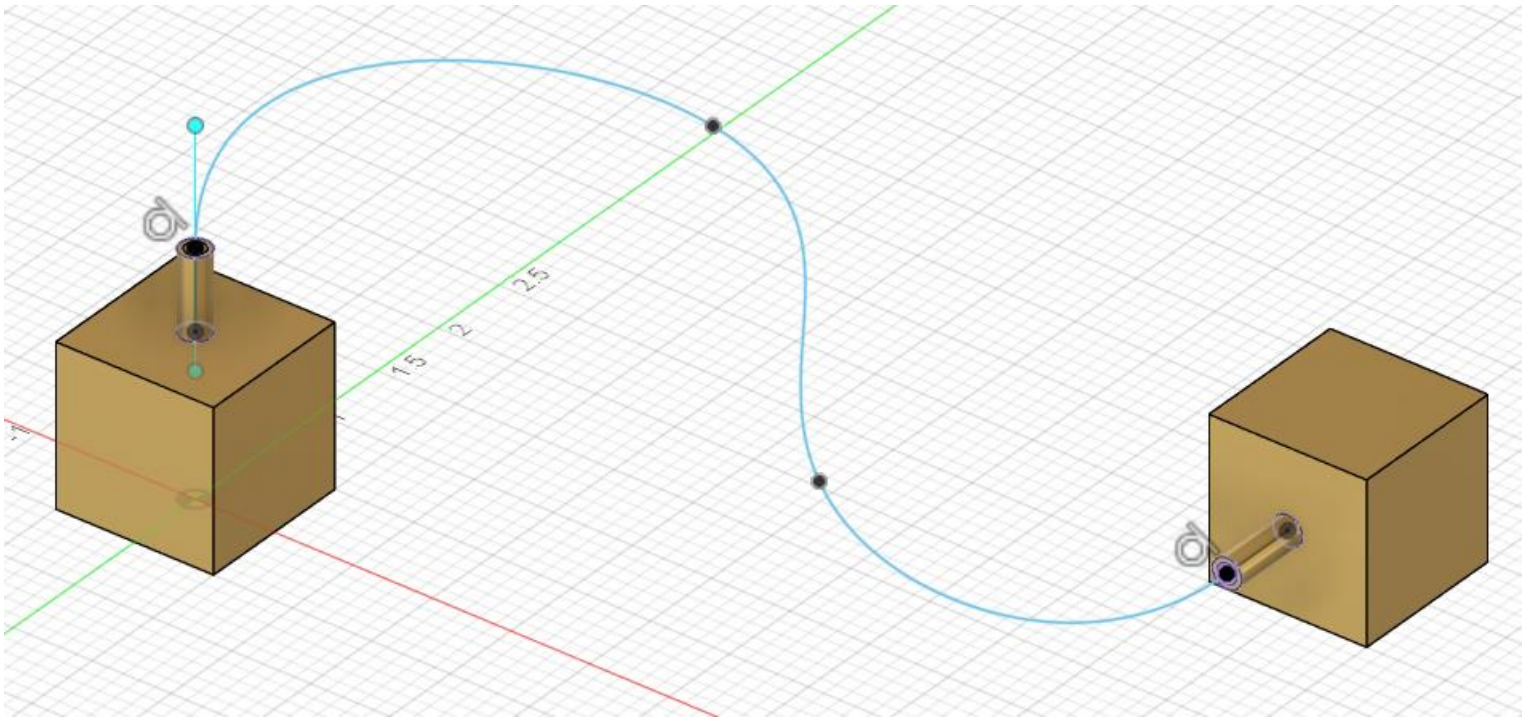- click on the **straight line** and then click on the **spline curve**

Select other geometries

BROWSER

JoeBarbetta_Hose v1

▷ Document Settings
▷ Named Views
▷ Origin
▲ Bodies
    Body1
    Body2
▲ Sketches
    FittingsGroup
    Fitting1
    Fitting2
    HosePath

The part of the curve should straighten in-line with the tube line and the **Tangent symbol** should appear.

- zoom out and the end of each tube extrusion should have a **Tangent** constraint symbol
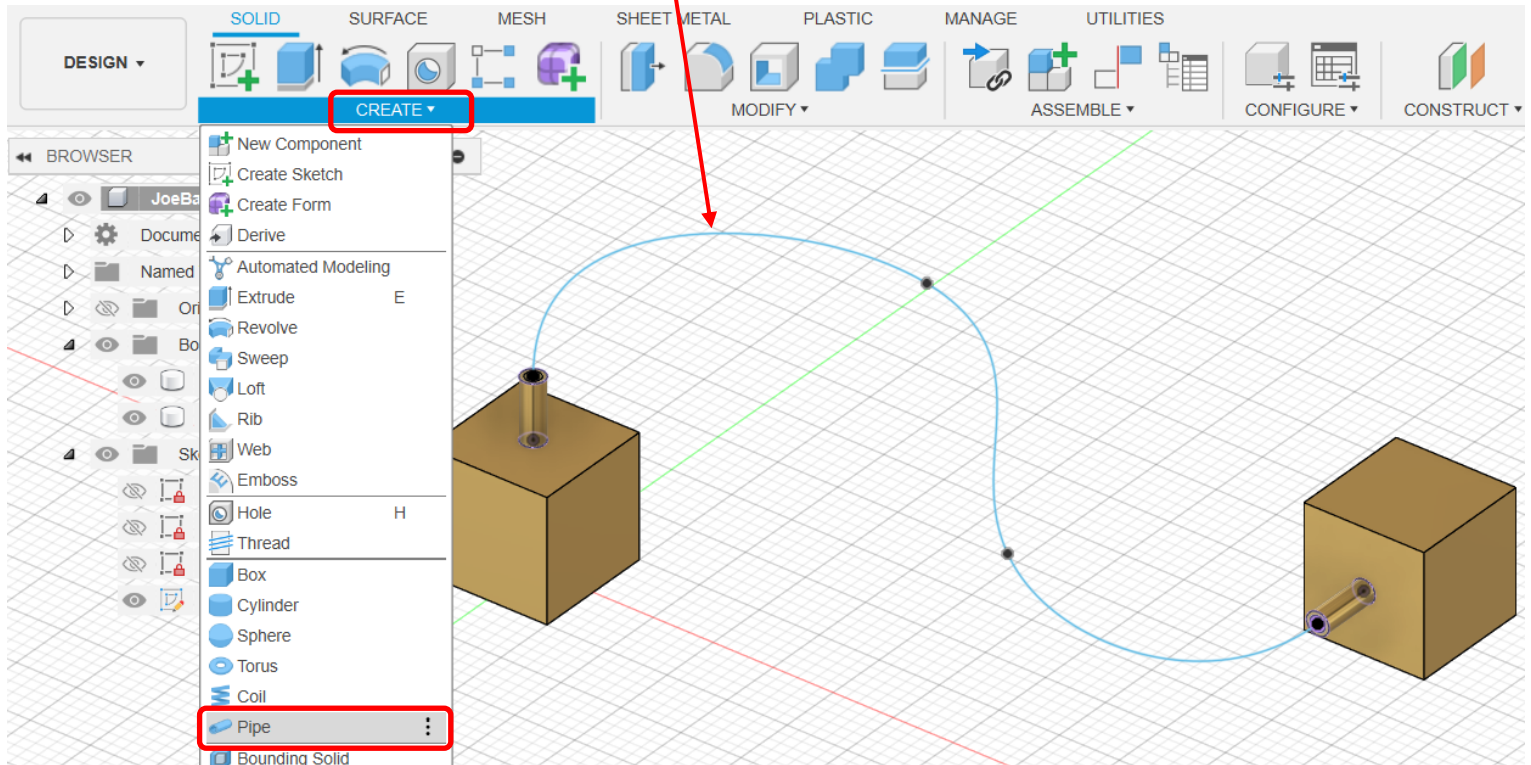
As mentioned before, your spline curve at this point may look very different, especially if additional points were added. Maybe you had too much fun and you turned the spline into an art project or perhaps a full rollercoaster track. Isn't there a game app where you create a crazy roller coaster and get the riders to vomit?
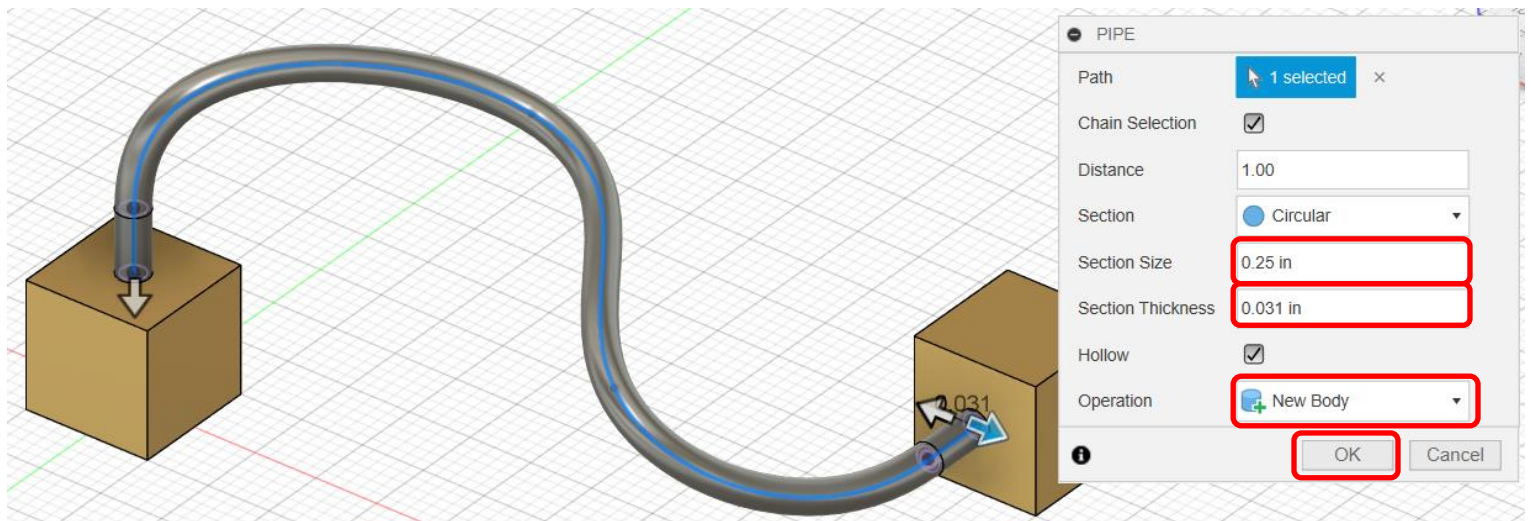
Creating a Pipe

Fusion has a **Pipe** tool that creates a pipe or tube that follows the line or curve on either a 2D or 3D Sketch to create a new body. Note that traditionally, one would construct a **Plane Along Path**, then create a sketch on that plane for the **cross-section** of the pipe or tube, and then use the **Sweep** tool. The **Pipe** tool is a short cut that combines these steps.

- from the **CREATE** menu select **Pipe** and click on the **Spline** curve
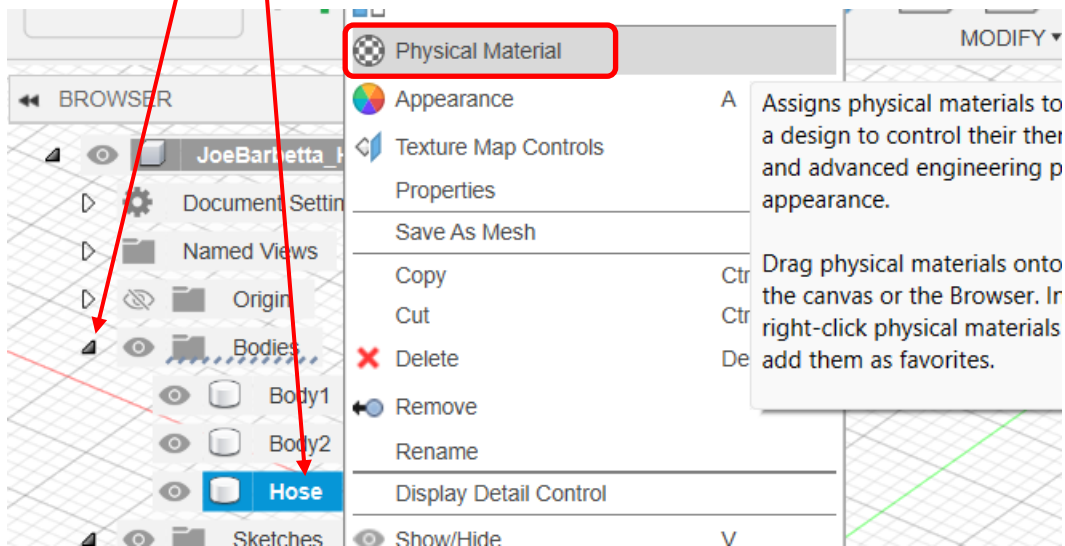

- set the **Section Size** to **0.25** and set the **Section Thickness** to **0.031**
- check the **Hollow** checkbox
- set **Operation** to **New Body** and click **OK**
- yell **"Dude, That's so neat!"**

The values used are for a common tubing size for a 3/16"ID (Inner Diameter) and 1/4" OD (Outer Diameter) tube, which would have a wall thickness of 1/32".
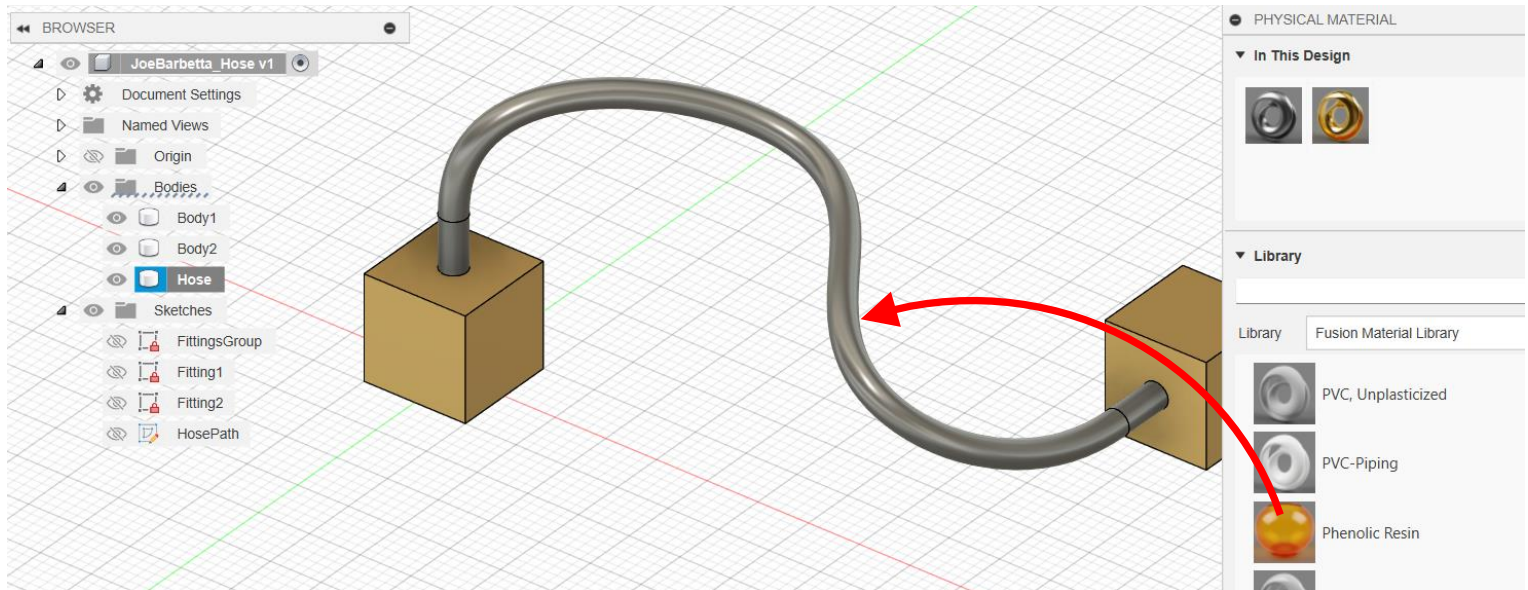
Setting the Physical Material

- click on the **arrow** for the **Bodies** folder
- rename **Body3** to **Hose**
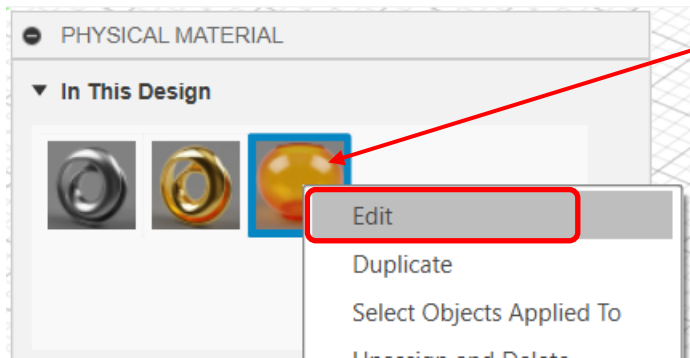- right-click on **Hose** and select **Physical Material**

- scroll down to the **Plastic** folder
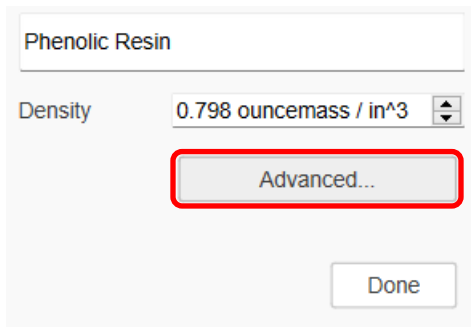- scroll down to **Phenolic Resin** and **drag its icon onto the tube**

Note that tubing would never be made from Phenolic Resin. Tubing is often made of Vinyl, which is primarily PVC with a high percentage of plasticizers. Phenolic Resin wa chosen because it is a translucent plastic in the library with a color.
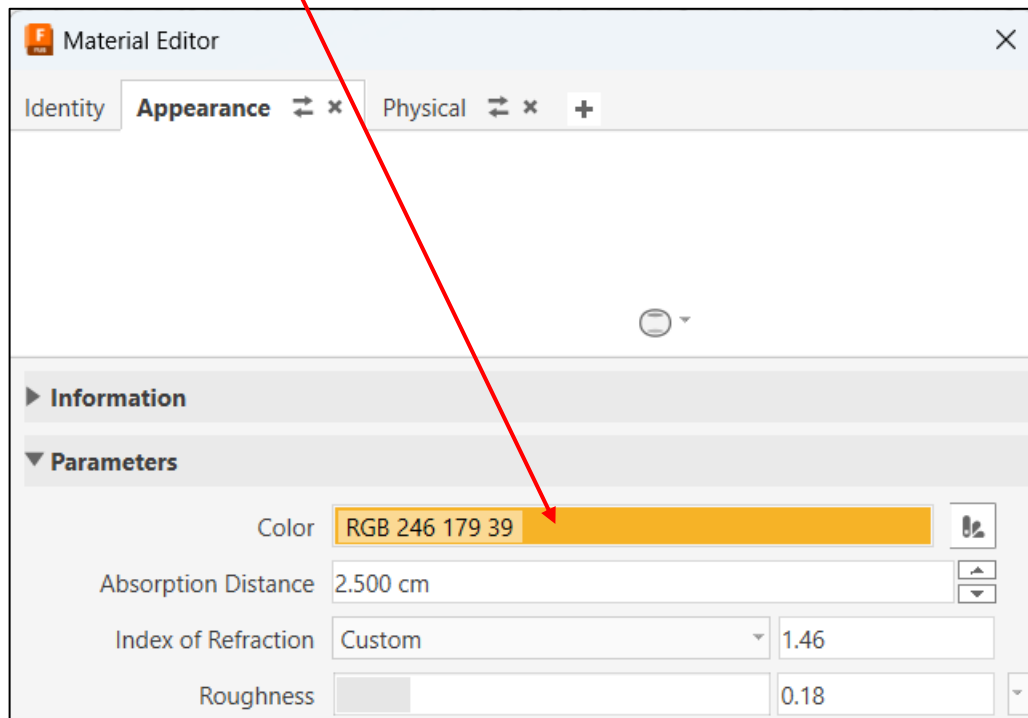
- at the top of the PHYSICAL MATERIAL window, right-click on the **Phenolic Resin icon** and select **Edit**
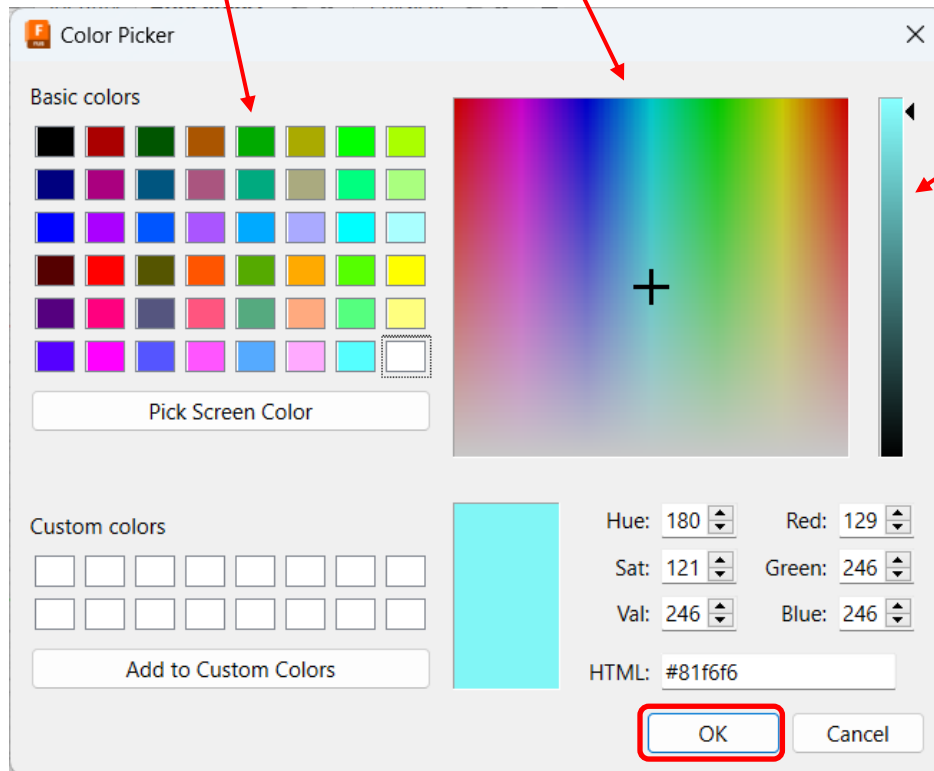

- click on **Advanced**

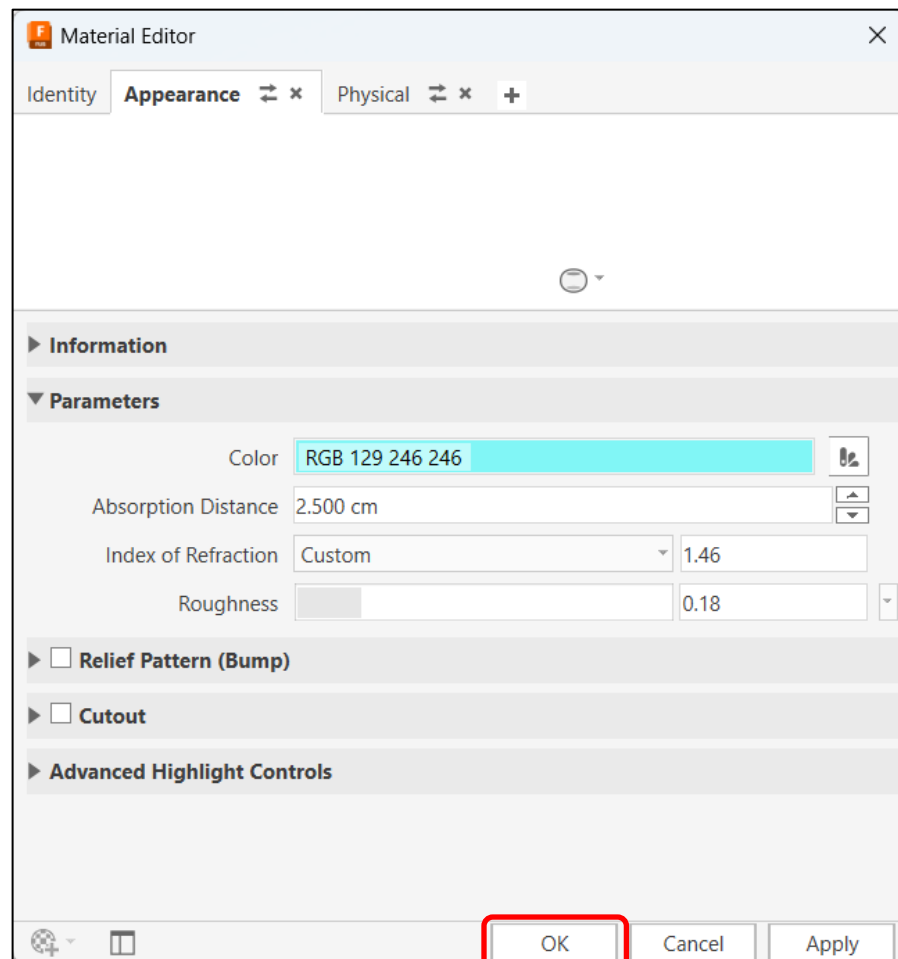- double-click on the **Color** box

- click on a **Basic color** box or on the **Color Gradient** box to select any color that you want. The right side **Brightness** bar can be used to lighten or darken the color.
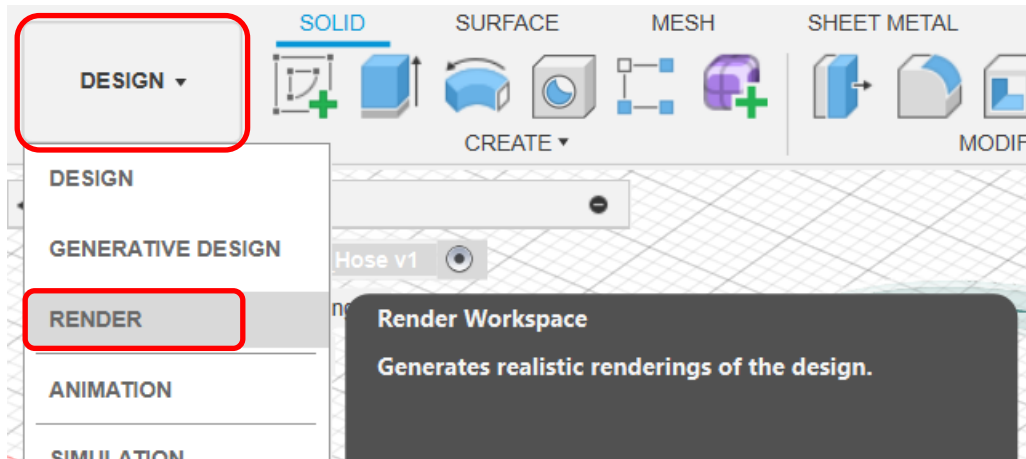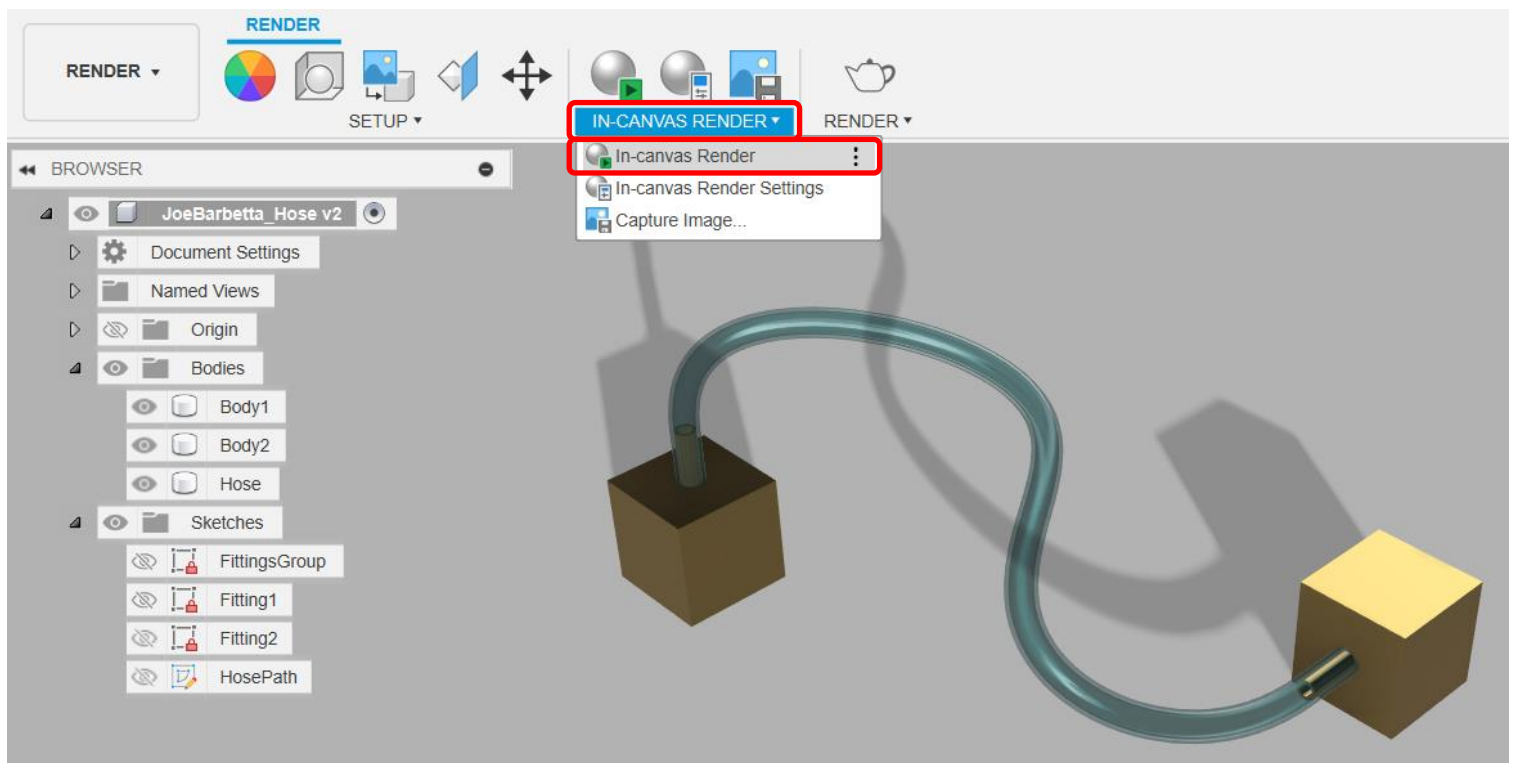- click **OK**

- click on **OK**

Using the Render Workspace

- using the **Workspace** box, select **RENDER**

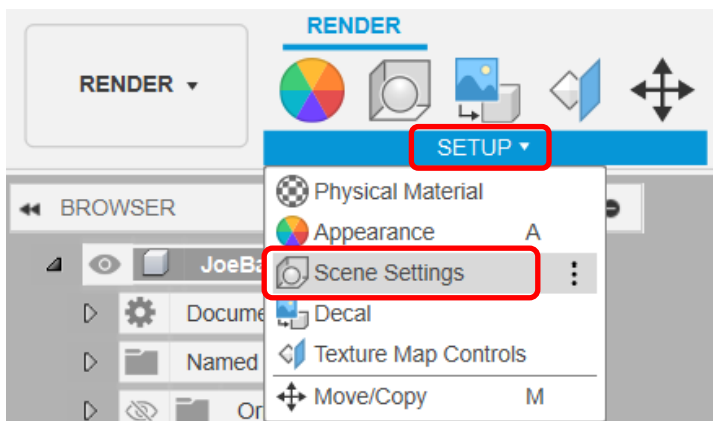- from the **IN-CANVAS RENDER** menu, select **In-Canvas Render**

- at the Progress bar at the bottom right of the Fusion screen move the slider from under **Excellent** to under **Final**

If renders are taking too long, for example, more than 2 minutes, the slider can be moved back towards **Excellent**.

Here is an example of a result.

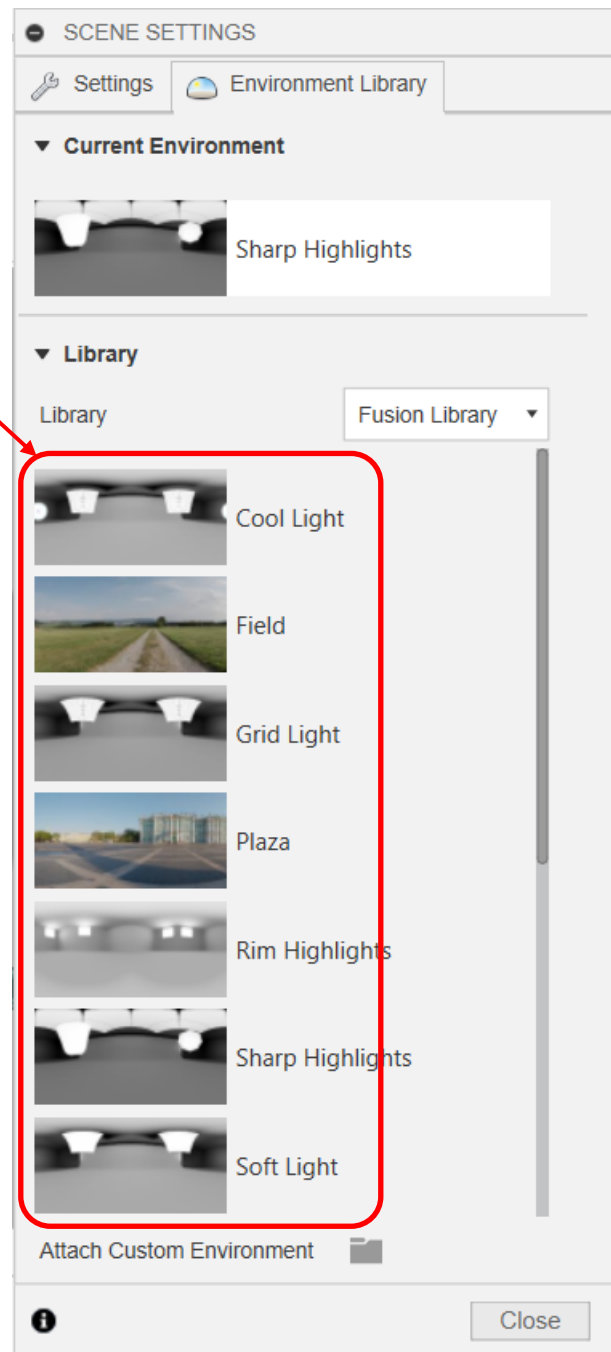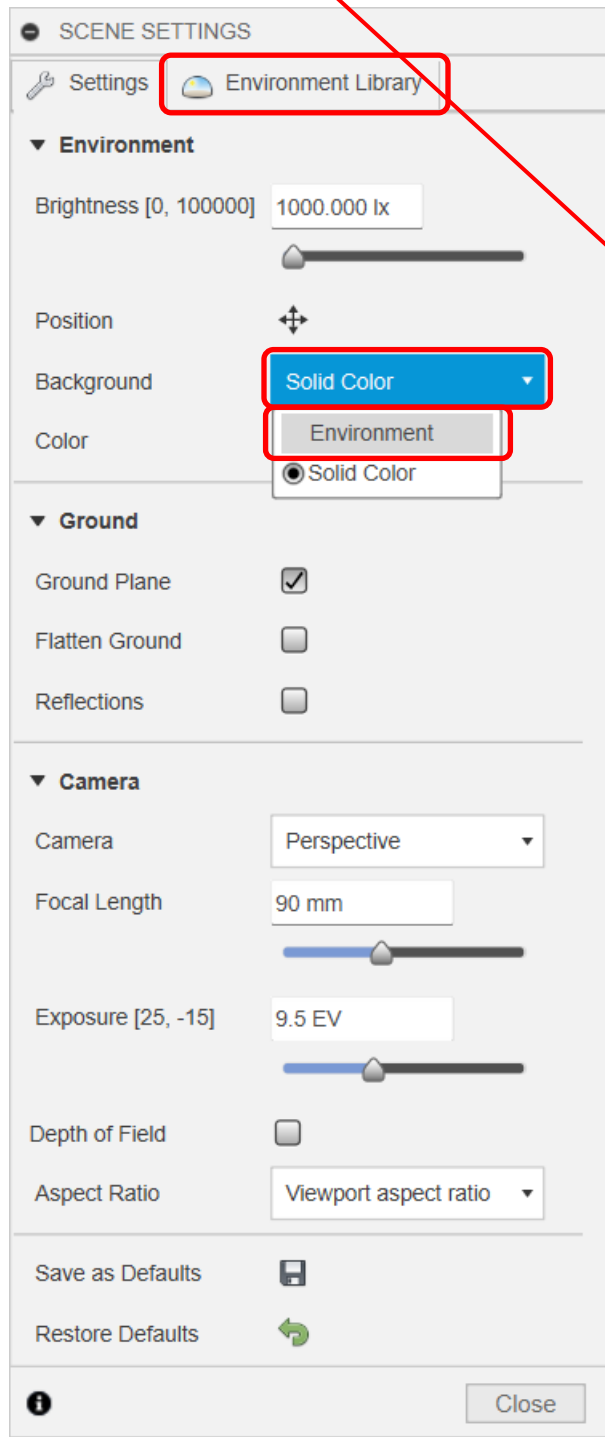- from the **SETUP** menu, select **Scene Settings**

One can click on the color bar under Solid Color or optionally, **Environment** can be selected.
- click on the **Environment Library** tab, which will show the options on the right.
- double-click on an **Environment** option to select it.

- run the render again and try other rendering settings until you get a result that you like and take a screenshot