

Autodesk Fusion Getting Started tutorial series

Video 9

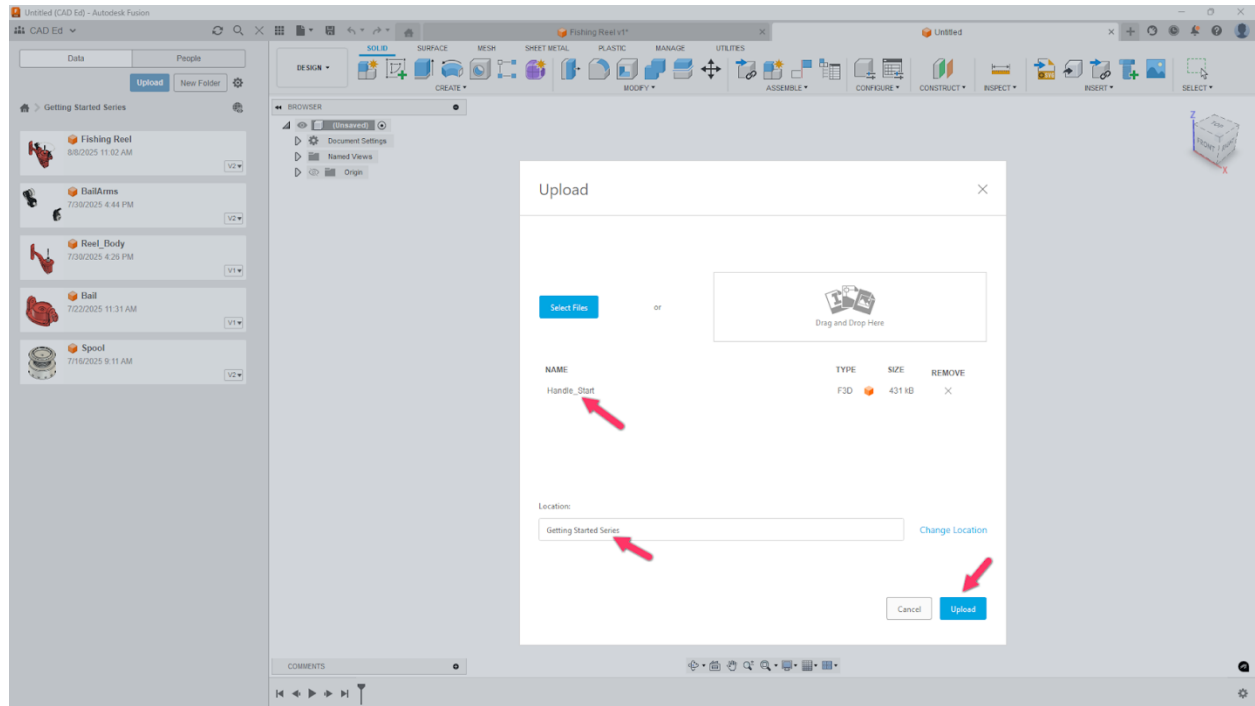


Exhibit 1

In this exhibit, we are uploading the Handle_Start file into the Getting Started Series project. Once the file is uploaded, open the file into Fusion.

File download:

[Handle_Start.f3d](#)

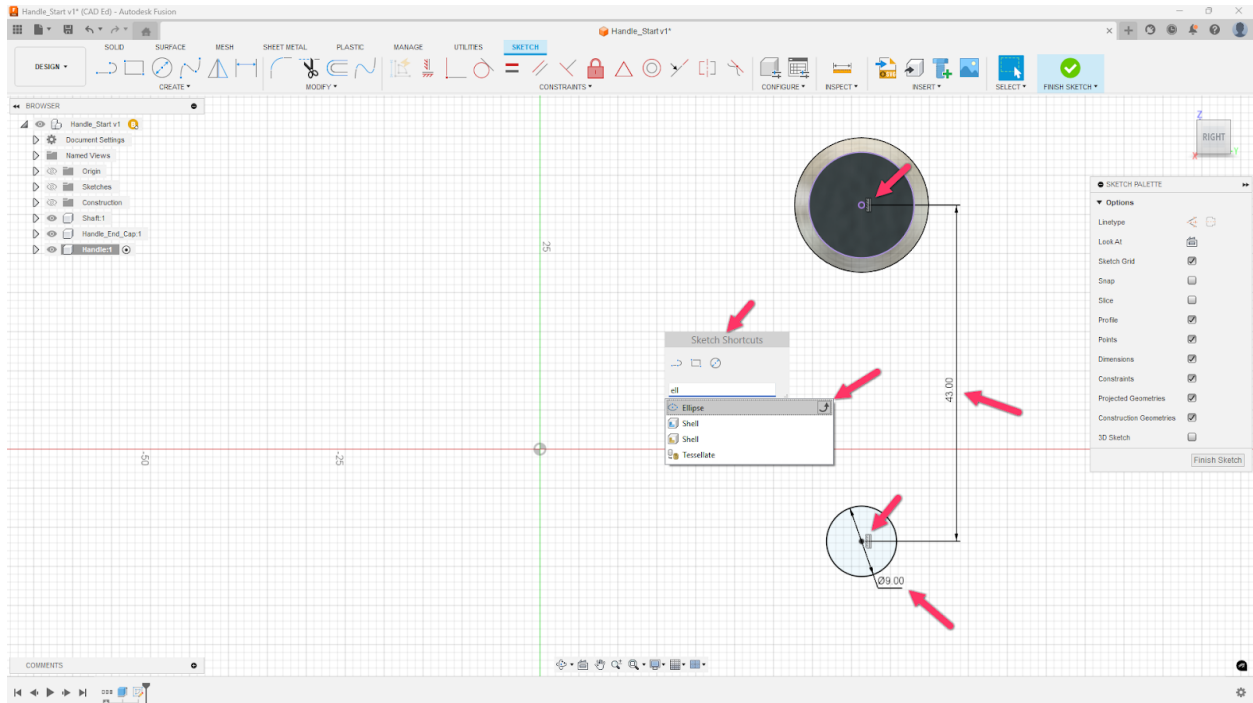


Exhibit 2

In this exhibit, we have projected the circular face of the black handle component. Then, a 9mm circle was drawn and vertically constrained with the projected circle. Dimension the circle to be 43mm below the projected circle.

Also, in this exhibit, you can see the S key shortcut brings up the Sketch Shortcut dialog. We have started typing in Ellipse and can press the up arrow next to Ellipse to add it to the Sketch Shortcut dialog for future use.

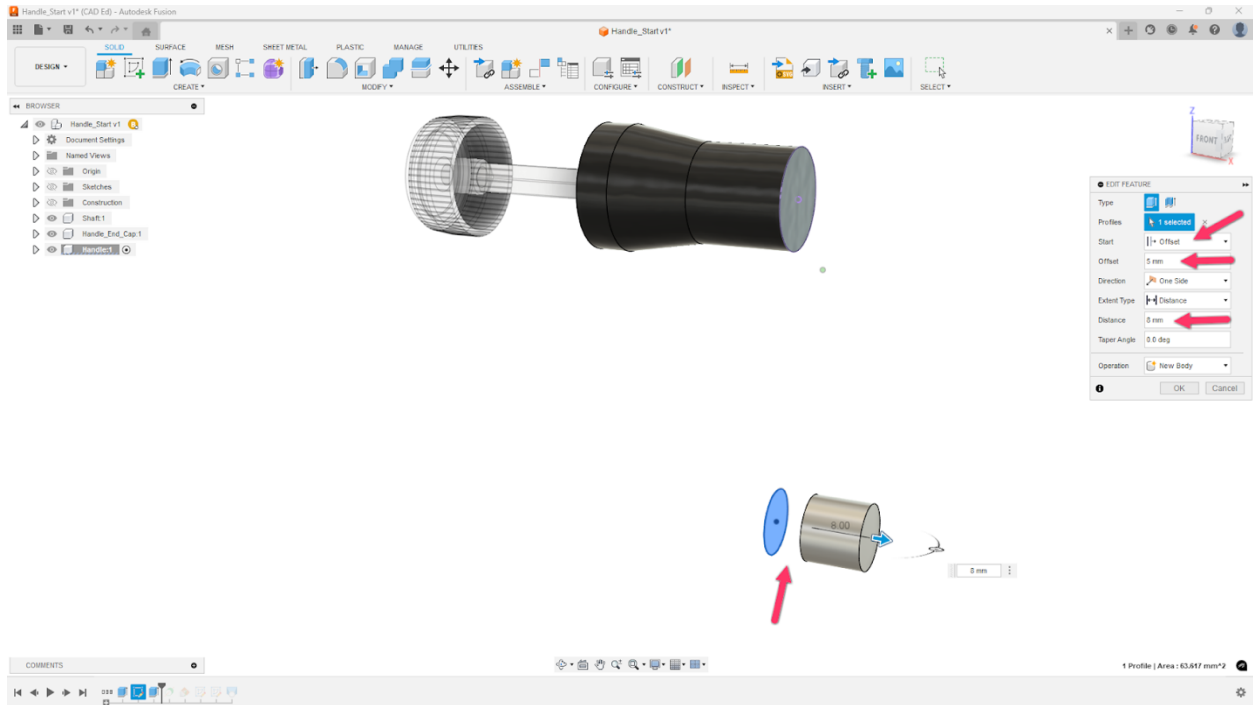


Exhibit 3

In this exhibit, we are using the Extrude command, but have changed the Start option to Offset and have set the Offset Distance to 5mm. The extrusion distance is set to 8mm, so the cylinder will be 8mm long, but created 5mm away from the sketch plane.

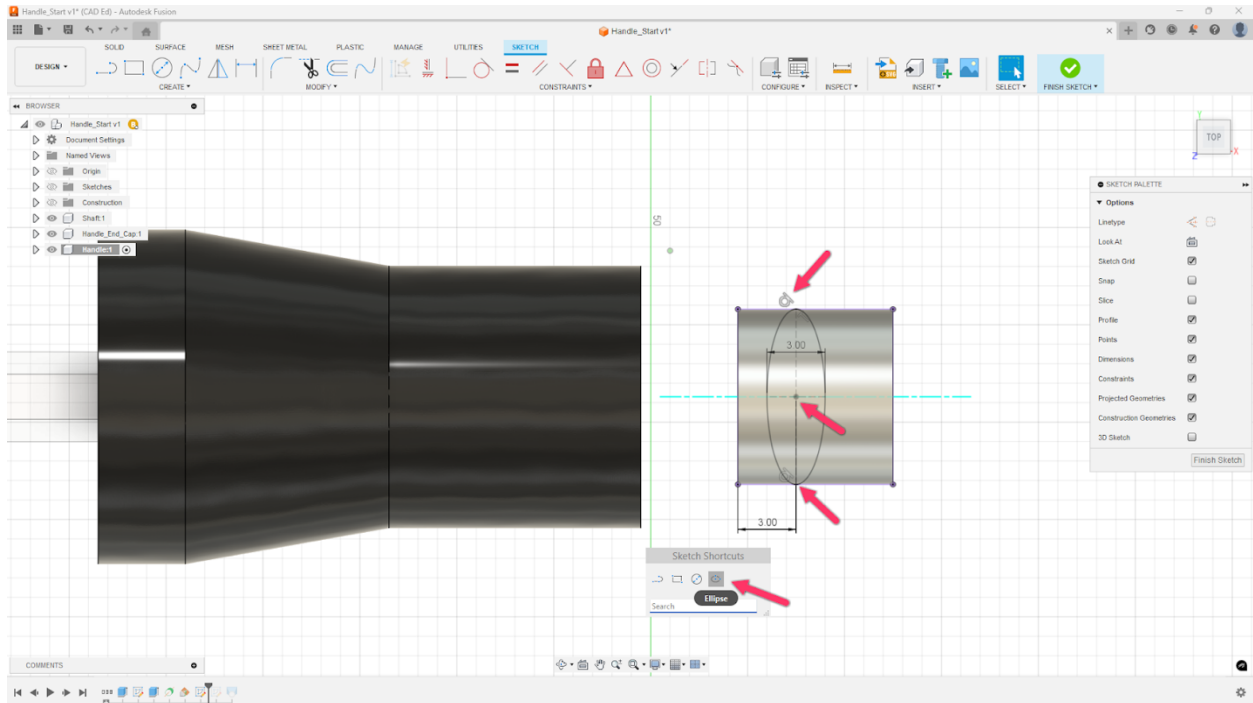


Exhibit 4

In this exhibit, we have projected the cylindrical body onto the sketch and then used the Ellipse command from the Shortcut Menu to create an ellipse on the axis. Snap the top of the ellipse to the top of the cylinder and then enter 3mm for the width. Dimension the center of the ellipse to be 3mm from the left edge of the cylinder. Constrain the top and bottom of the ellipse with Tangent Constraints to the sides of the cylinder.

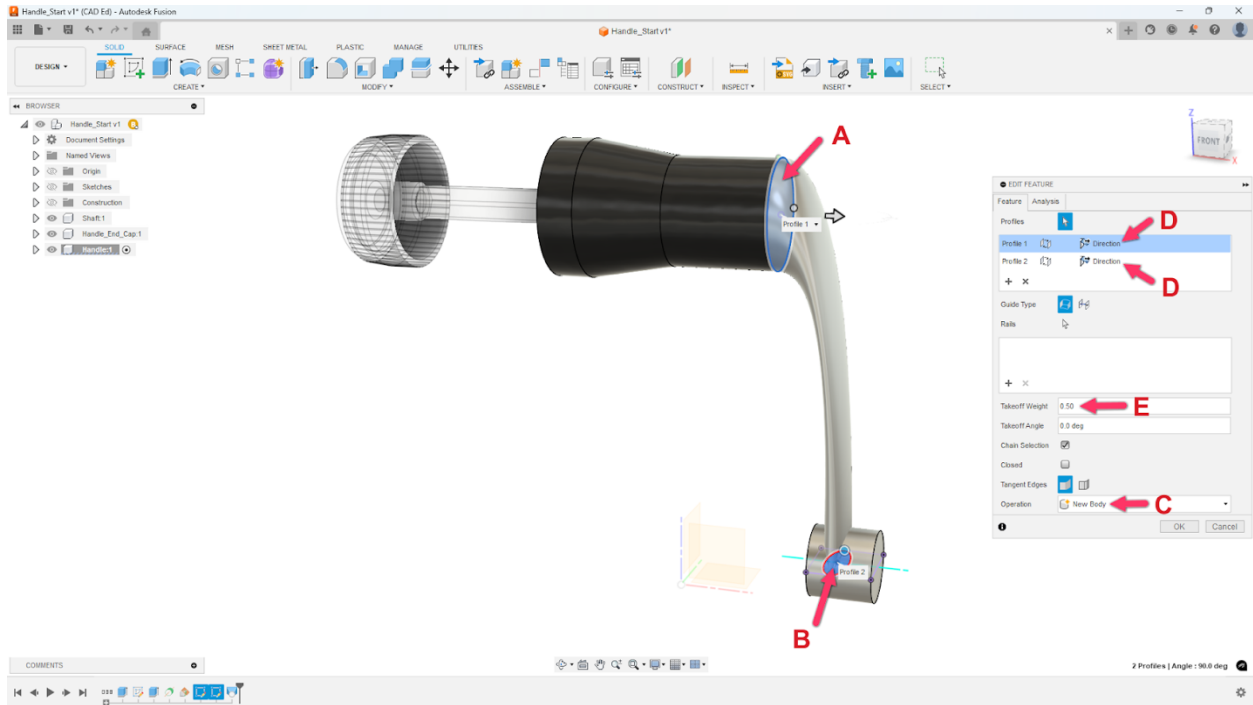


Exhibit 5

In this exhibit, we are in the Loft command. Start by selecting the two Profiles, A & B. The preview will probably turn red as the handle is cutting through the lower cylinder. Change the Operation to New Body (C). To give the handle a more natural look, change the Profiles in the dialog from Connected to Direction (D). Lastly, select Profile1 in the dialog and drag the Takeoff Weight arrow in the viewport to around 0.5. You can type 0.5 in the dialog to be precise.

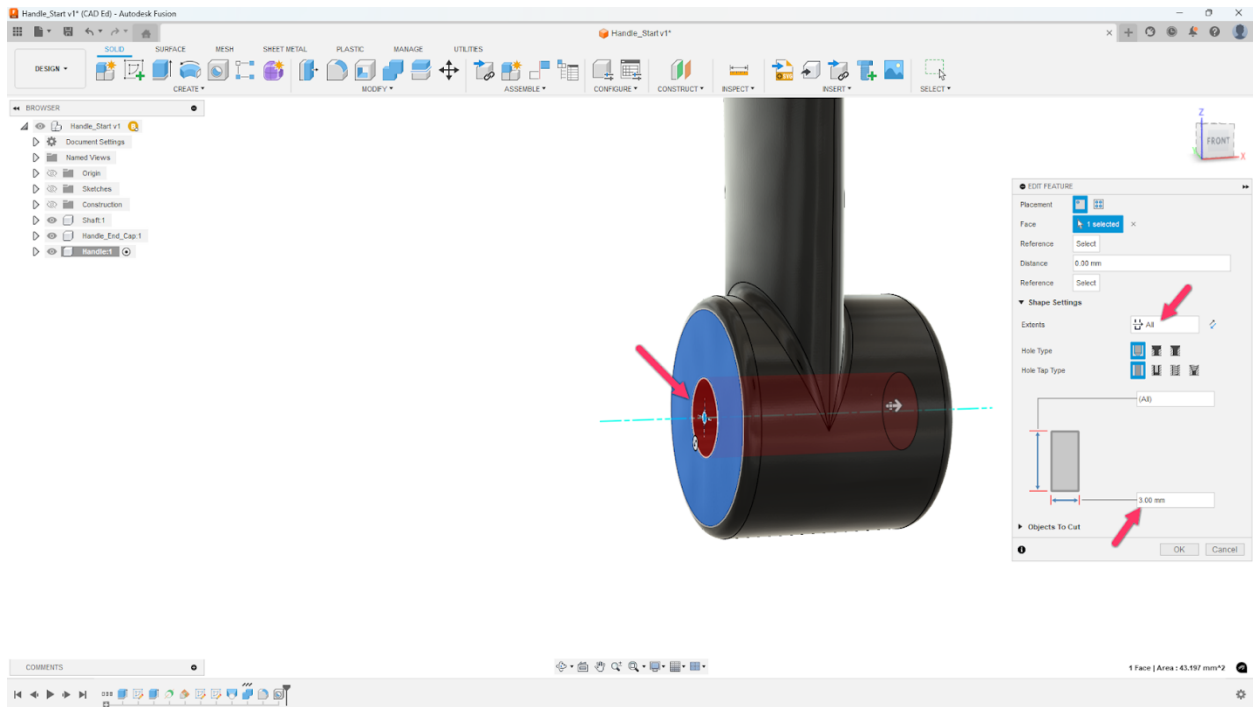


Exhibit 6

In this exhibit, we have added 0.5mm fillets to the 4 edges on the cylinder. Then, use the Hole command to create a 3mm hole that extends all the way through the cylinder.

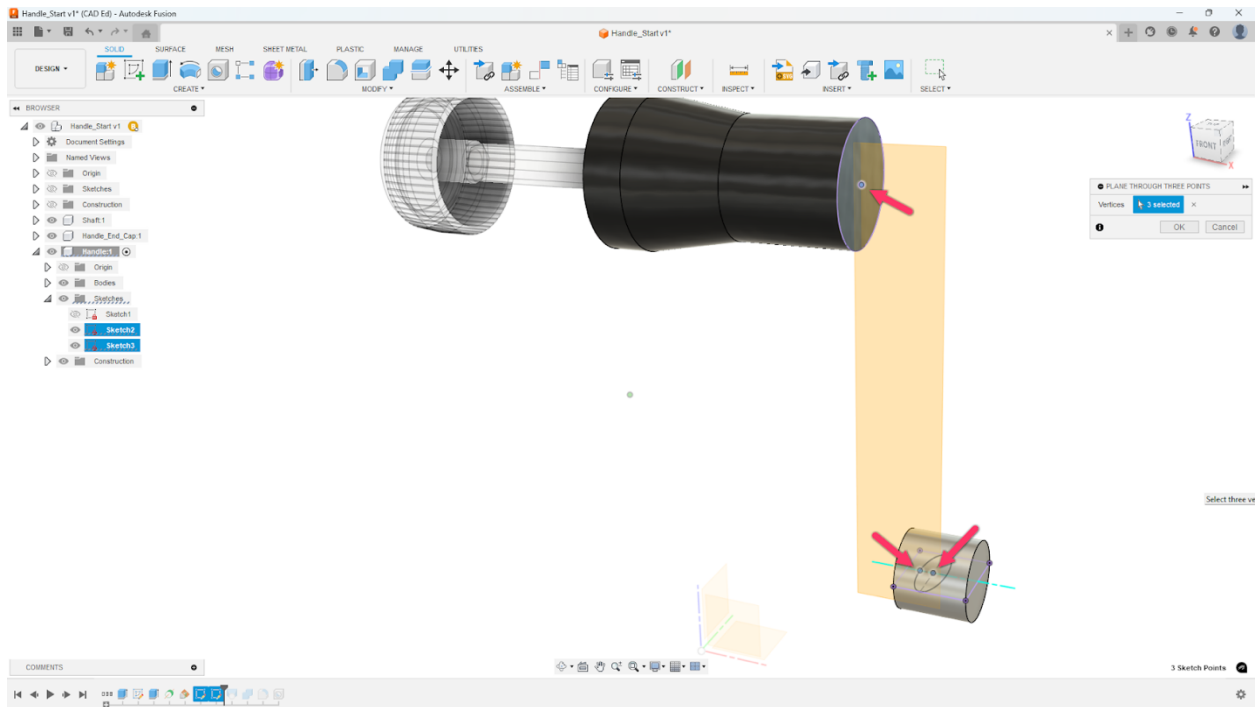


Exhibit 7

In this exhibit, we have turned on Sketch2 and Sketch3 in the Browser. Then, create a Plane Through 3 Points and select the three points as shown in the exhibit to create a plane that slices through the three points.

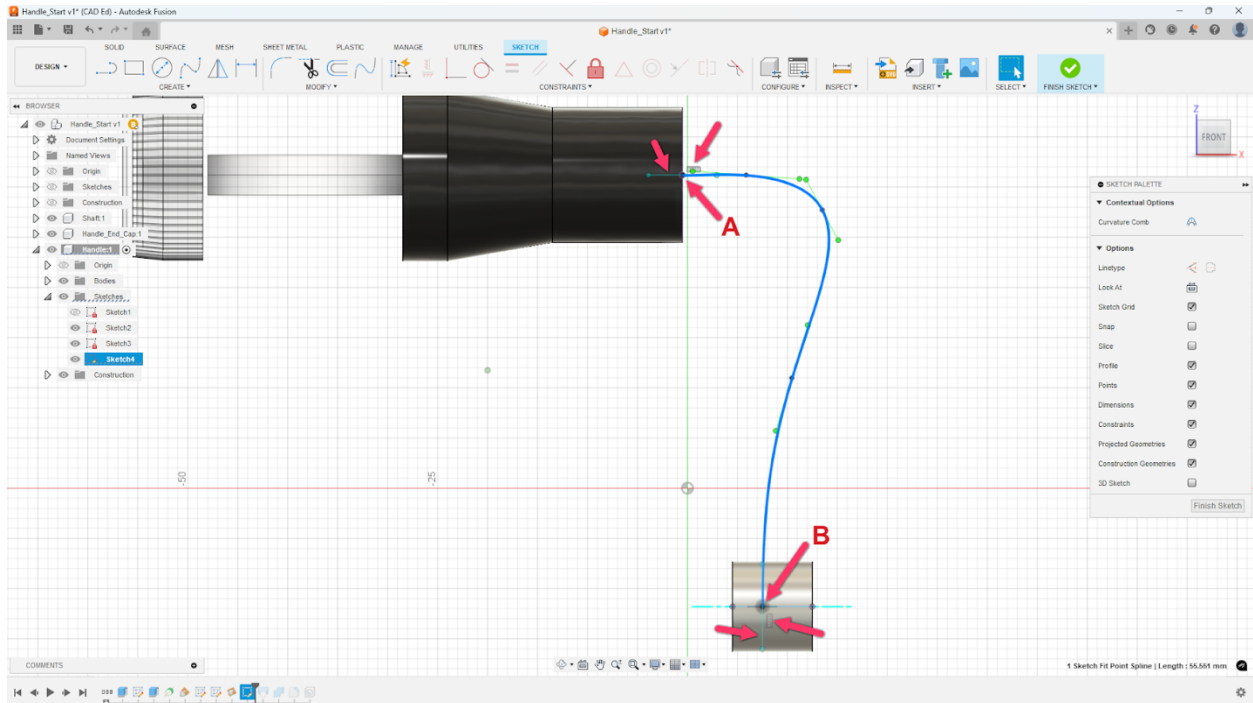


Exhibit 8

In this exhibit, we have projected the points A and B onto the sketch. Using the Fit Point Spline command, start at Point A and draw a question mark type spline down to Point B. Finally, use the Horizontal/Vertical Constraint and select the green tangent line near Point A to make it horizontal and near Point B to make that green tangent line vertical.

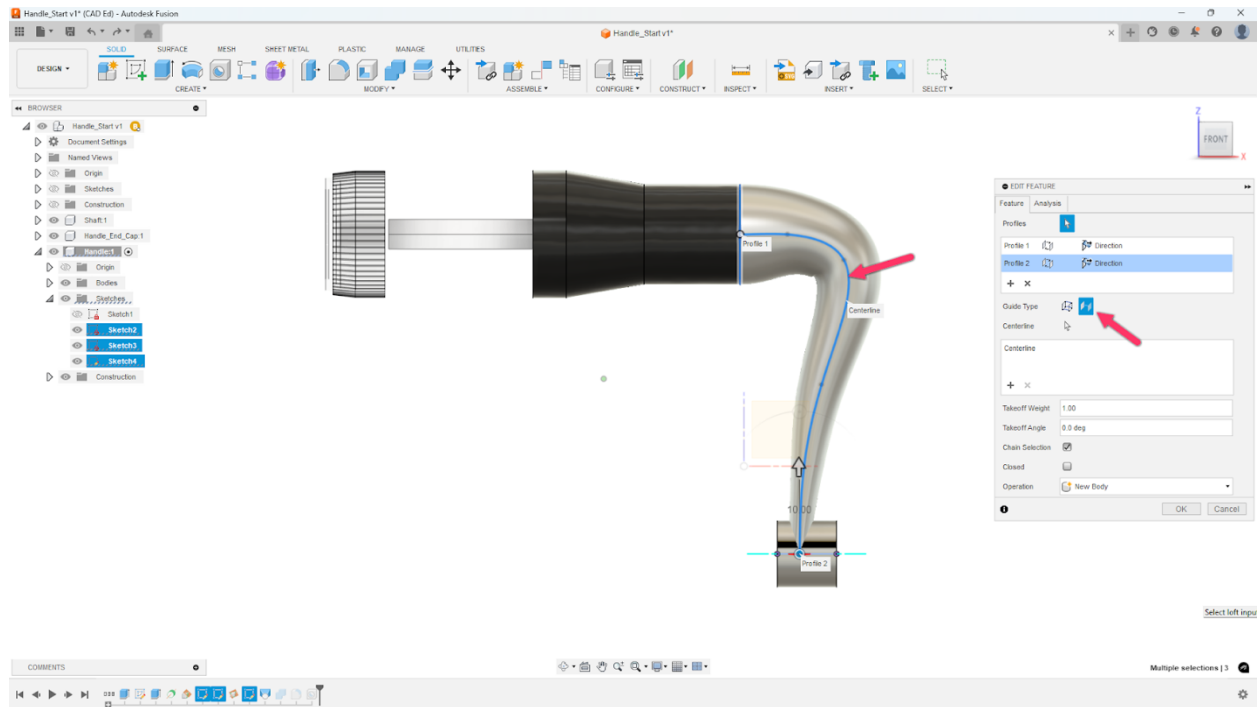


Exhibit 9

In this exhibit, we have edited the Loft feature in the timeline and selected the Centerline Guide Type and selected the spline as the Centerline.