

Autodesk Fusion Getting Started tutorial series

Video 13 – Inspection

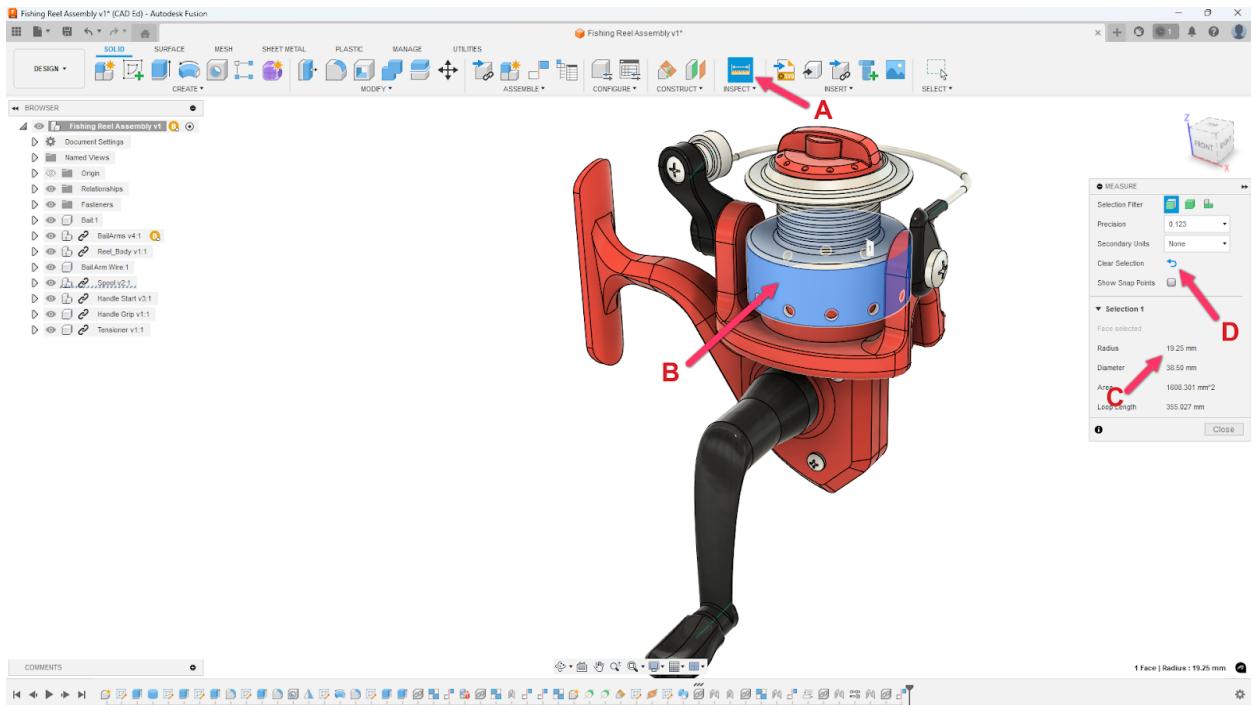


Exhibit 1

In this exhibit, we have clicked on the Measure command (A) and then selected the cylindrical face of the spool (B). The Measure Dialog shows the Radius and Diameter (C) of the face you selected. Then click on Clear Selection (D) to take another measurement.

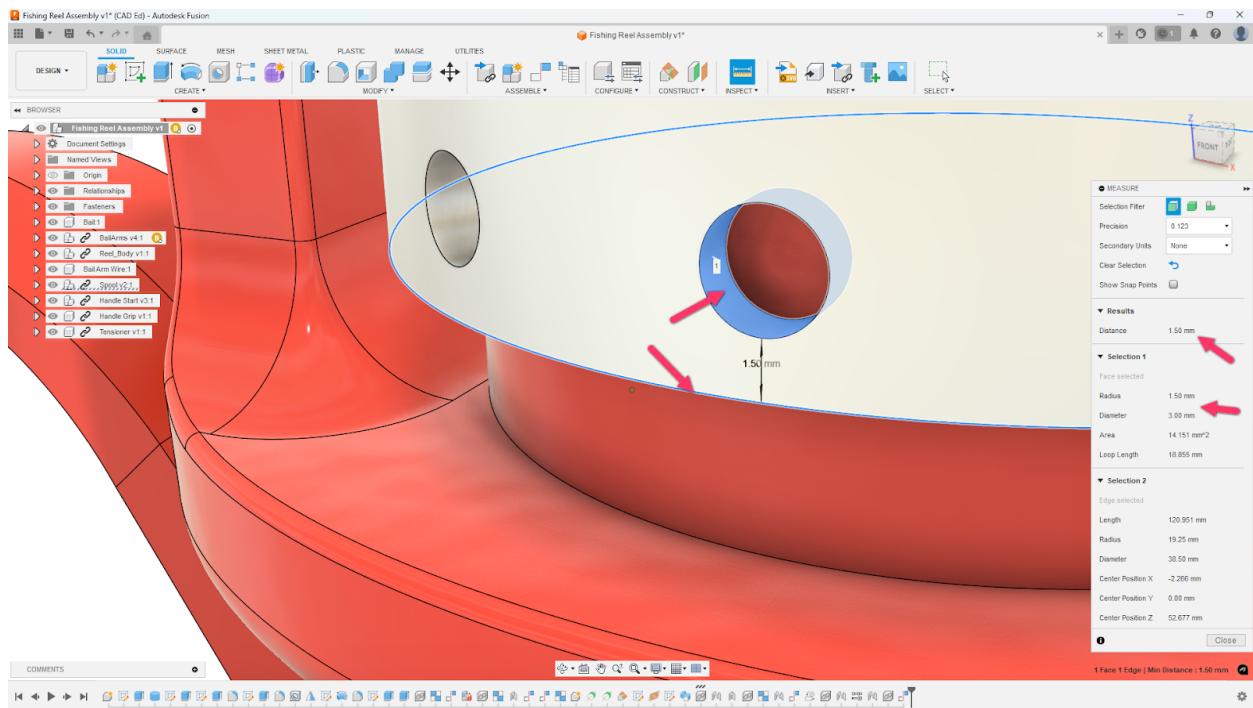


Exhibit 2

In this exhibit, we have selected both the cylindrical face of the hole and the bottom edge of the spool. The Measure Dialog returns the hole radius and diameter along with the shortest distance measurement between the hole and the edge.

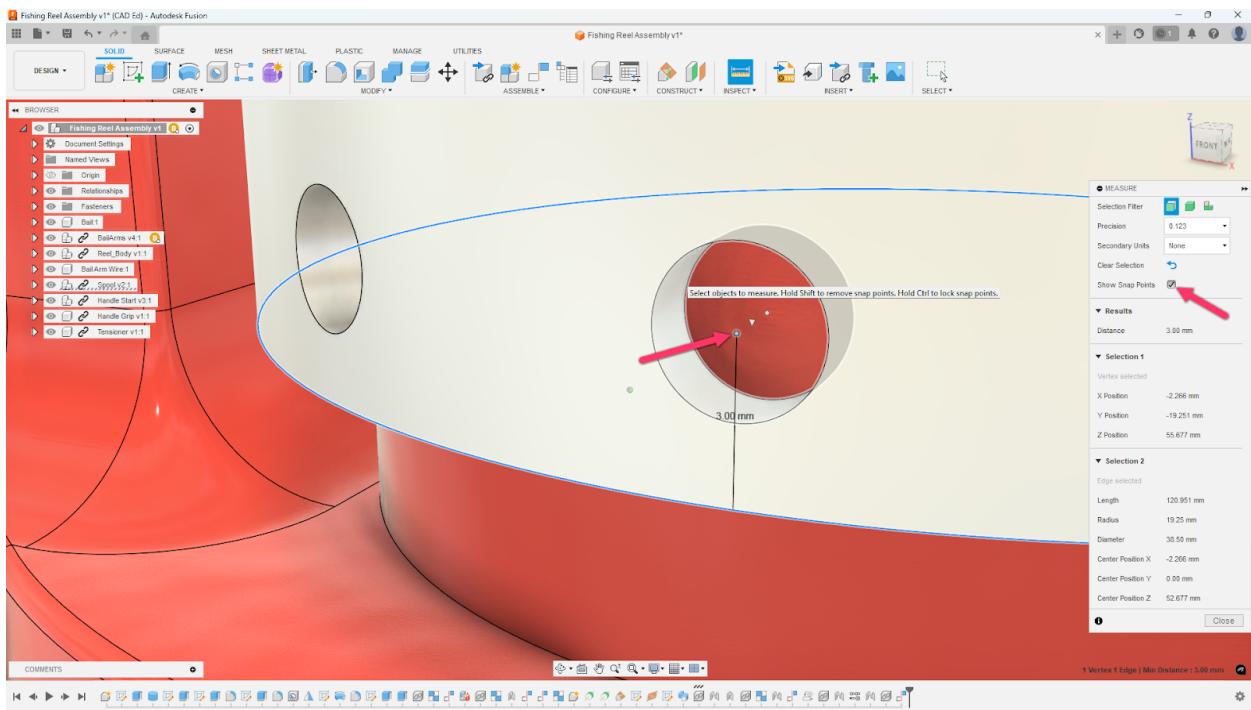


Exhibit 3

In this exhibit, we have turned on the Show Snap Points option in the Measure Dialog. Then, click on the cylindrical surface of the hole and it will snap to the center of the hole. Click on the bottom edge of the spool and the Measure Dialog will return a distance of 3mm.

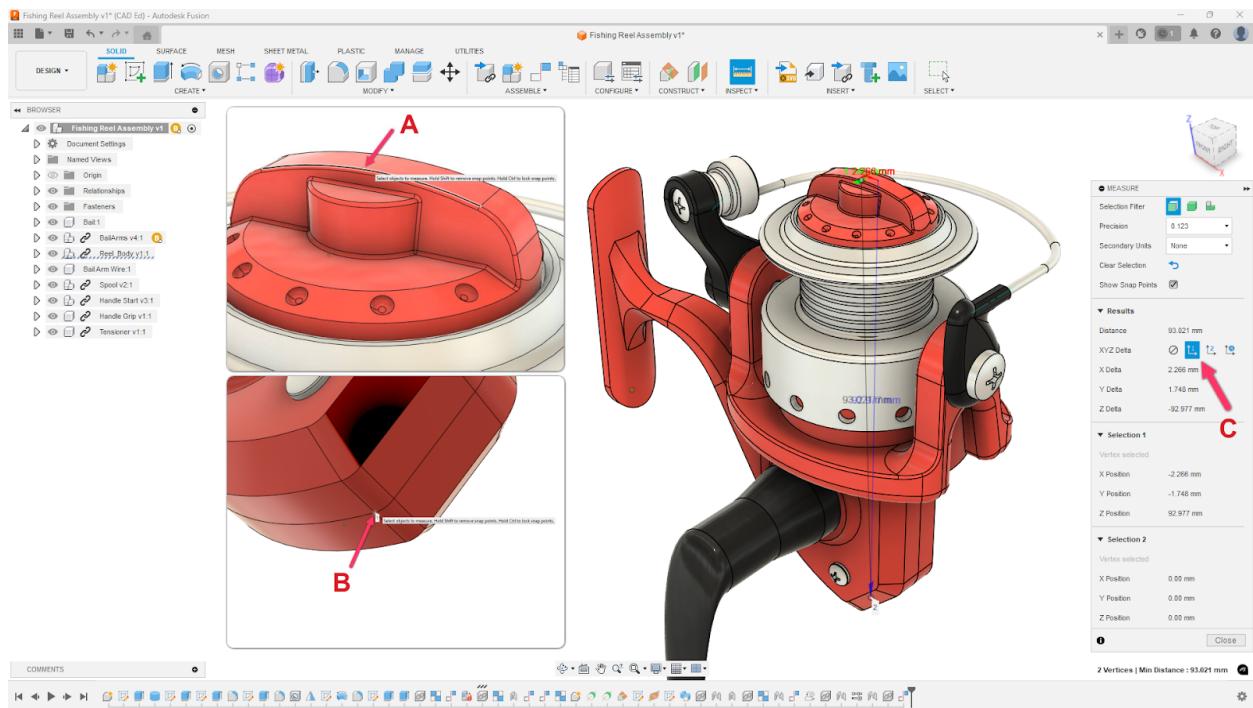


Exhibit 4

In this exhibit, we have selected the top-middle of the edge on the Tensioner (A) and the bottom middle point of the Reel (B). Then, in the Measure Dialog, select the XYZ Delta icon to display the X, Y, and Z delta distances. In this case, the total vertical distance is 92.977mm (Z Delta).

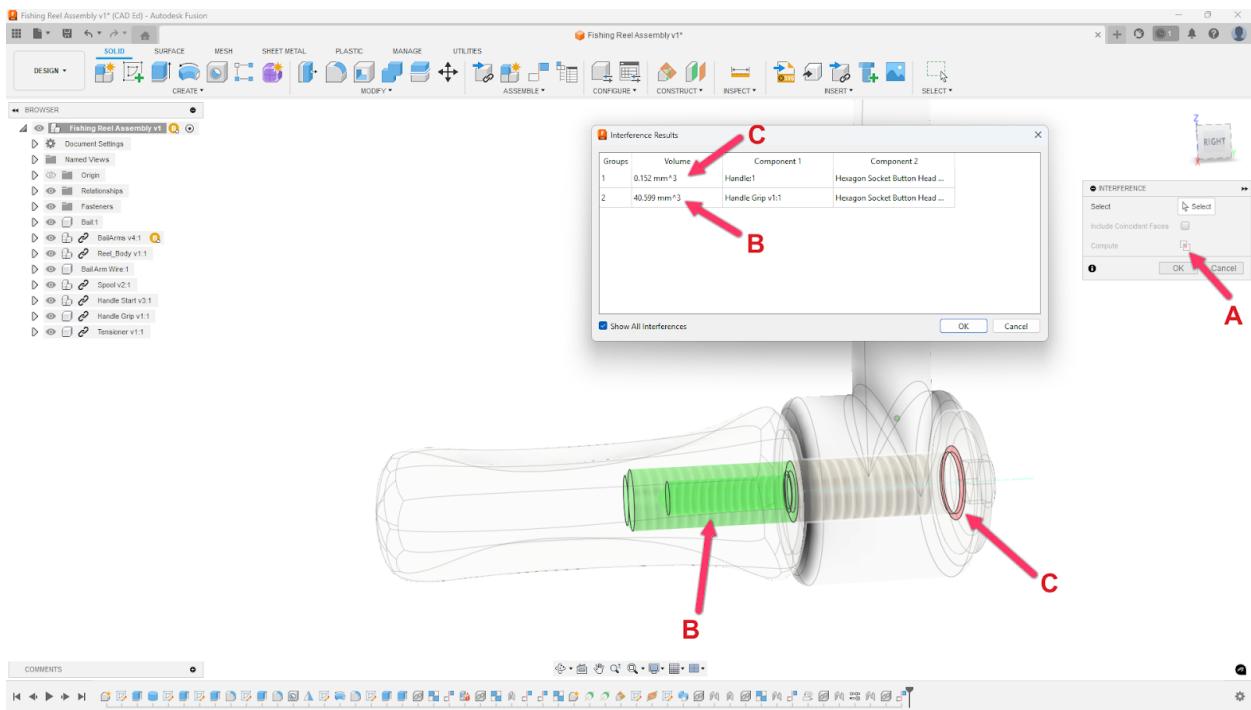


Exhibit 5

In this Exhibit, we have run the Interference command from the Inspect menu and selected the Handle and Handle Grip parts, then clicked on the Compute icon (A). The Results dialog appears showing two interferences (B & C). Clicking on a group in the dialog will highlight it in the viewport.

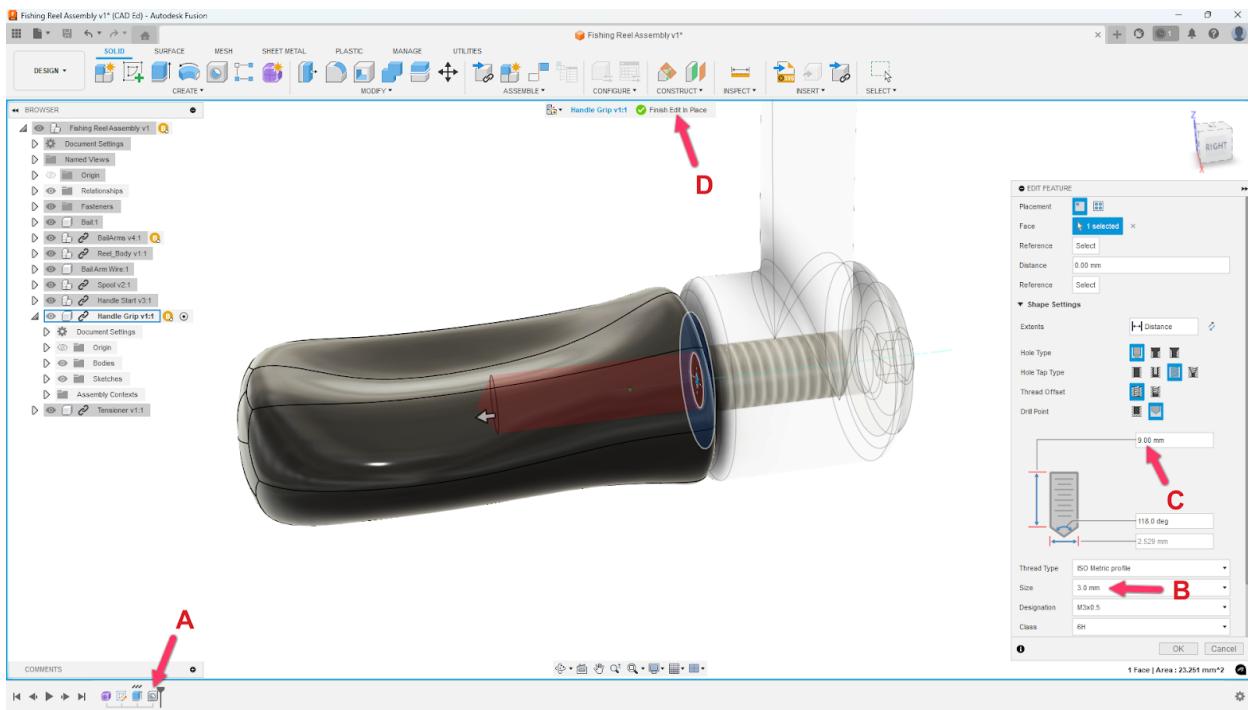


Exhibit 6

In this exhibit, we have entered the Edit In Place environment by clicking the pencil icon next to Handle Grip in the Browser. Then, edit the Hole feature (A) in the timeline. Change the Size to 3mm (B) and the Depth to 9mm (C), then click on Finish Edit In Place (D) to accept the changes to the design.

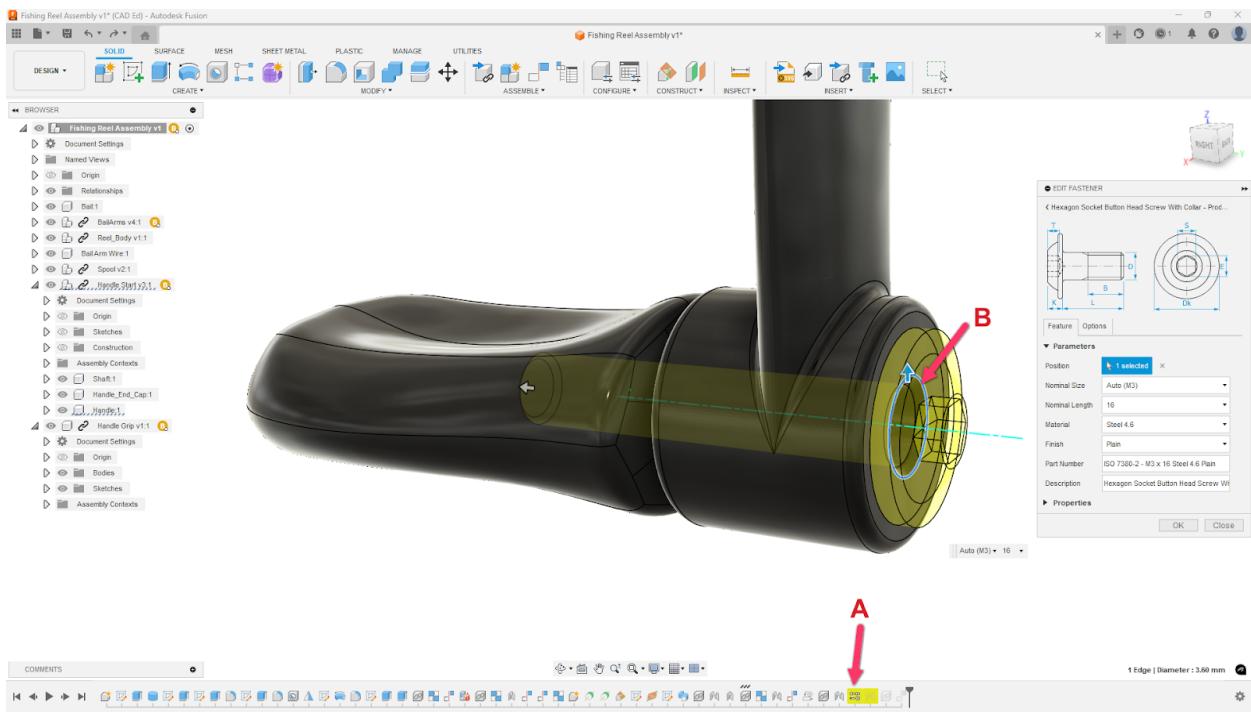


Exhibit 7

In this exhibit, we have edited the Fastener feature in the timeline (A) and selected the new edge (B) to redefine where the fastener should be referenced.

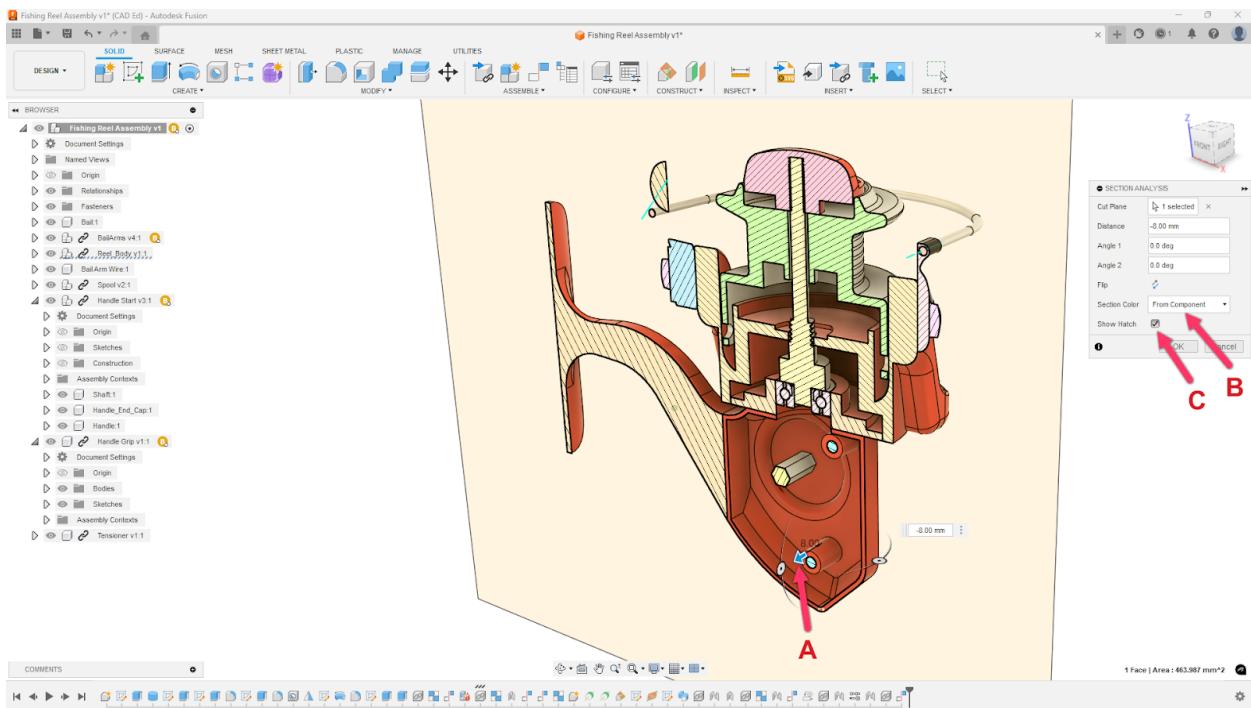


Exhibit 8

In this exhibit, we have used the Section Analysis tool to slice through the design. Drag the blue arrow (A) to change the depth of the section. Click on a cylindrical face to snap to the center of that cylinder. You can change the Section Color (B) or show/hide the Hatch Pattern (C).

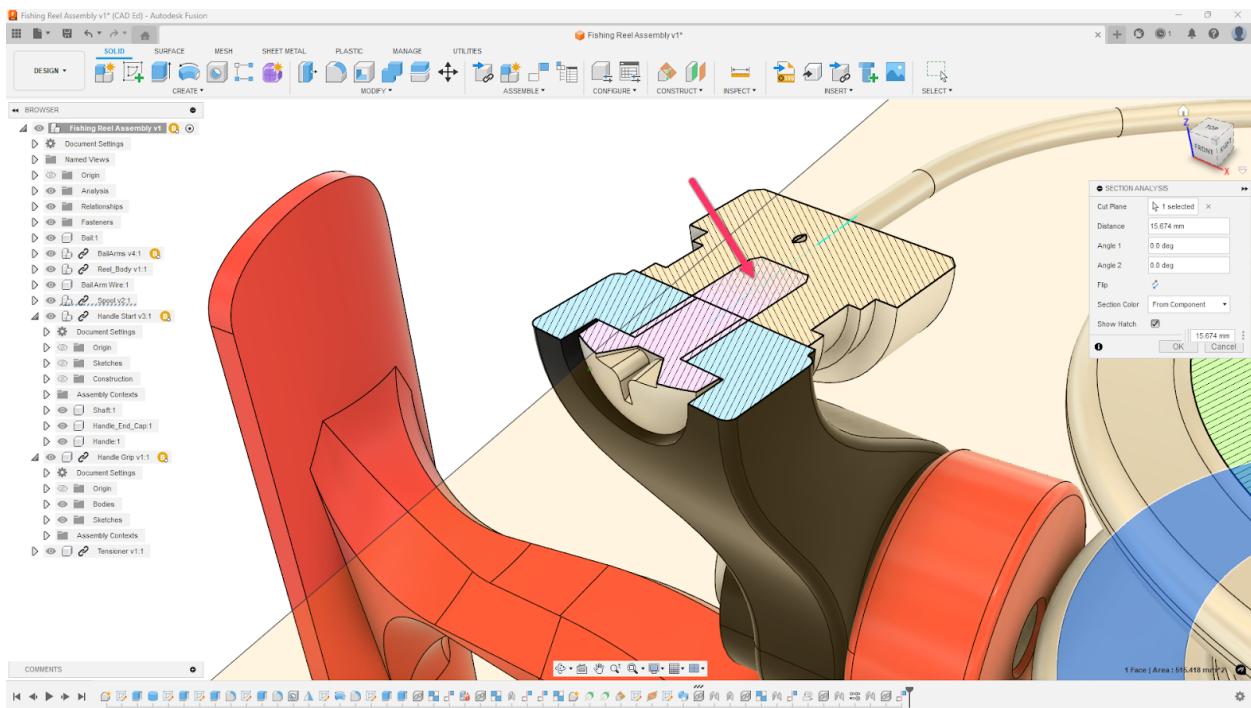


Exhibit 9

In this exhibit, we have created a section view that slices through the LineRoller component. We can see some cross-hatching going on, signifying there is clashing between the Screw and the LineRoller components. This means there is no hole in the LineRoller component for the screw. Press OK to accept the section view.

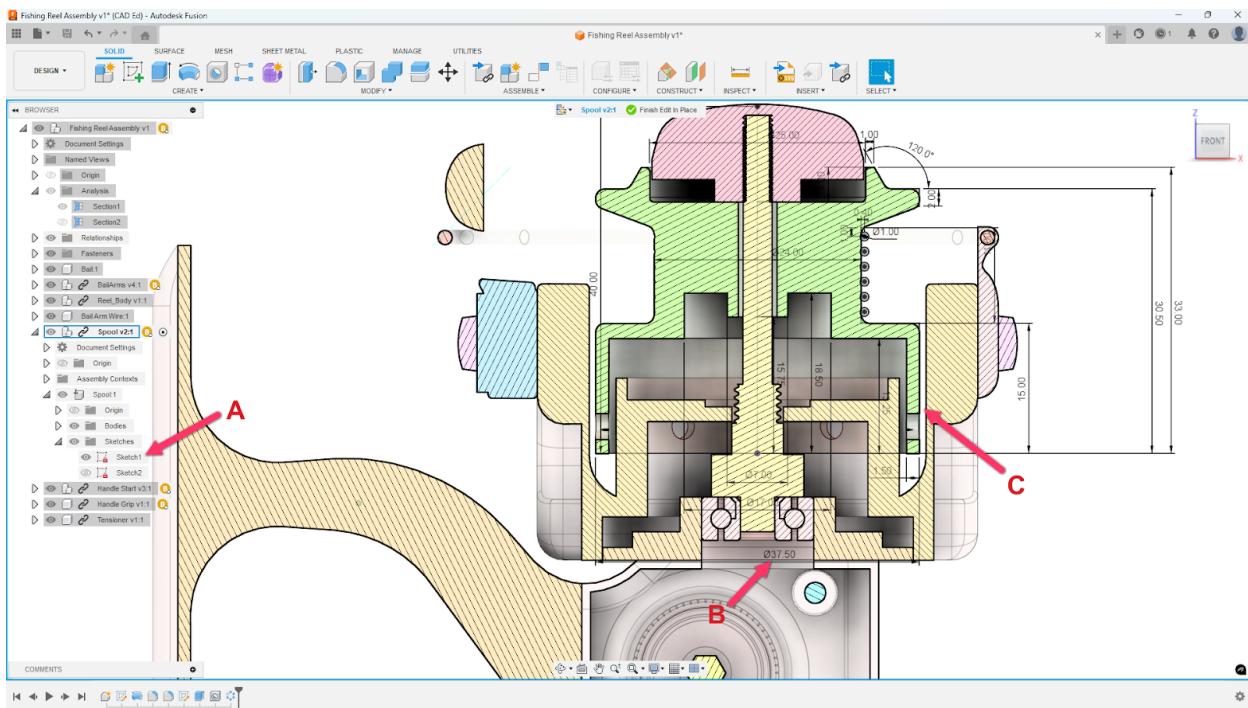


Exhibit 10

In this exhibit, we have done an Edit In Place of the Spool component and then turned on Sketch1 (A). Edit the 38.5 diameter dimension and change it to 37.5mm (B). Notice the spool updates accordingly and is easy to see the result in the Section View (C).

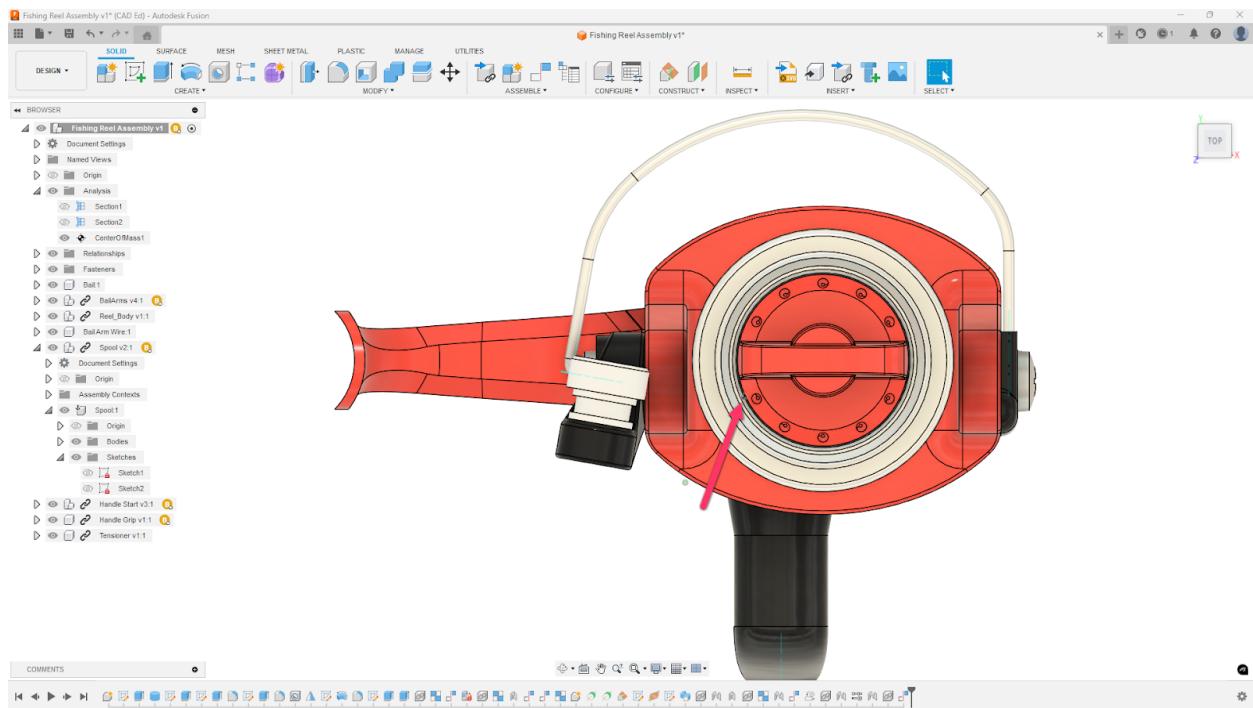


Exhibit 11

In this exhibit, we have done a Center of Mass inspection and selected the top-level assembly. By viewing the assembly from the top, we can see that the Center of Mass icon is not as centered in the assembly as we would have expected. This is due to the mass properties of the handle being incorrect.

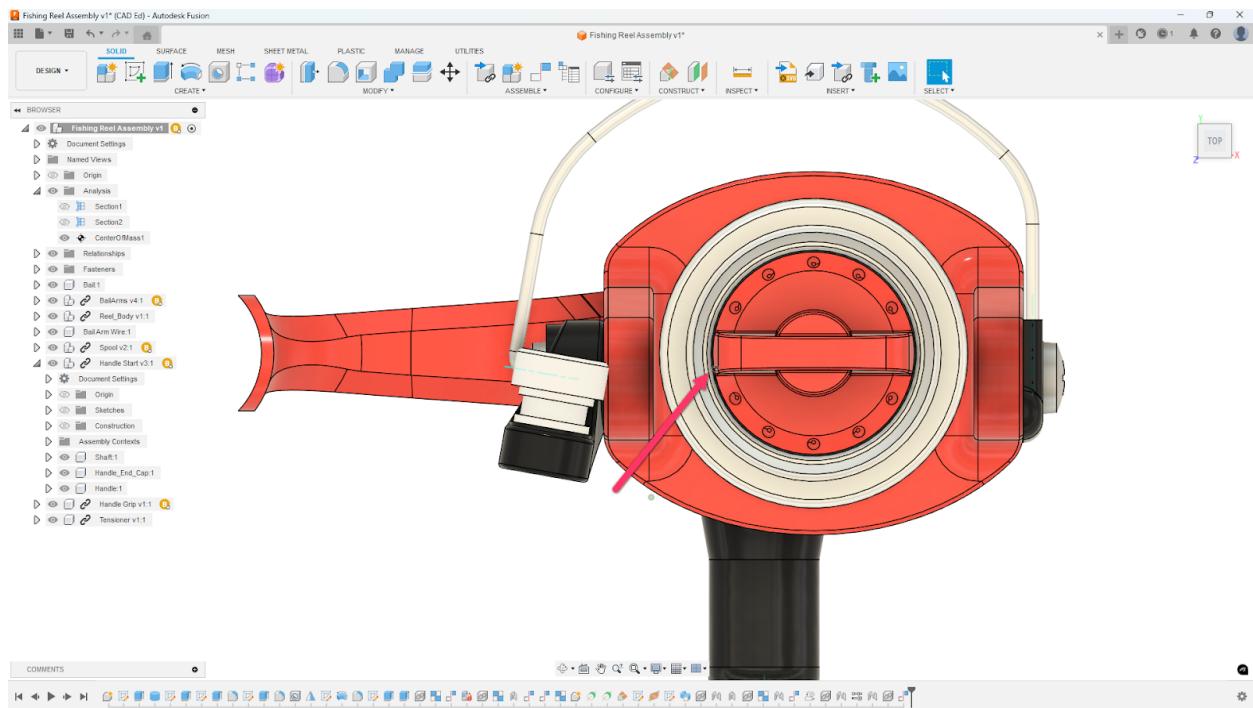


Exhibit 12

In this exhibit, we have changed the physical material of the handle to be ABS Plastic, which causes the Center of Mass icon to move more toward the centerline of the reel.