# Autodesk Fusion Getting Started tutorial series

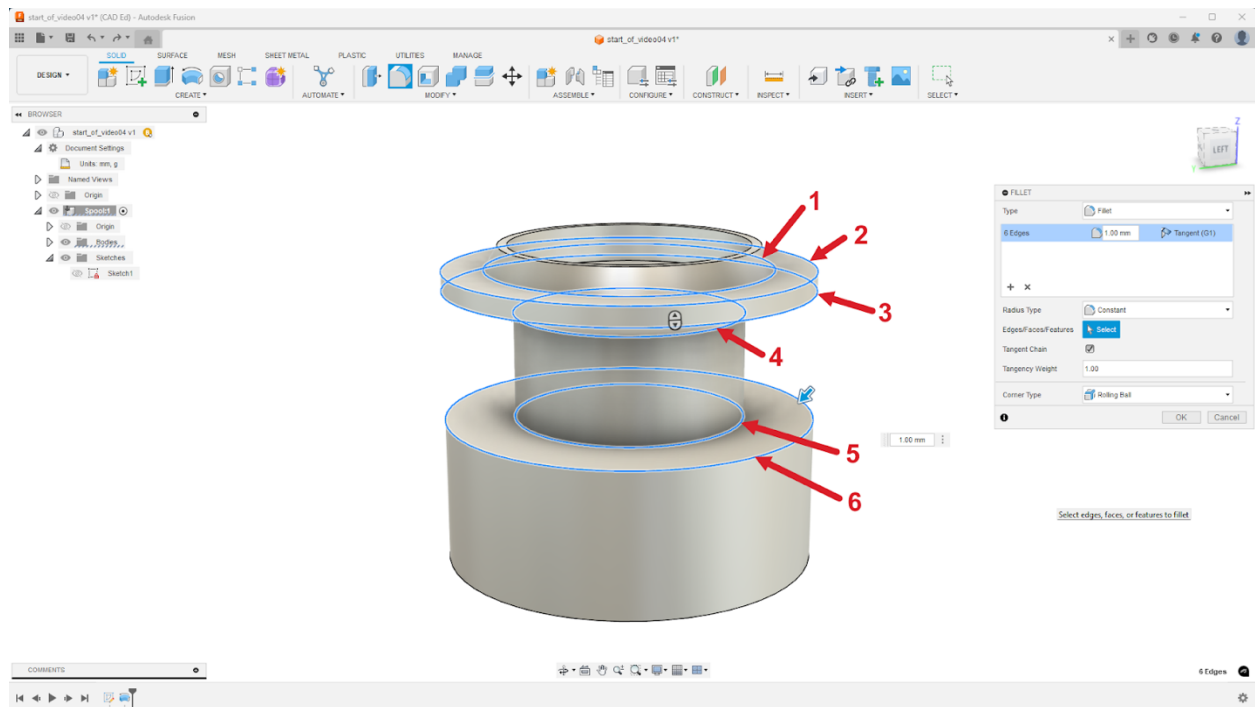## Video 4 – Creating a 3D Model



**Exhibit 1**

This exhibit highlights the 6 edges designated for a 1mm fillet application.
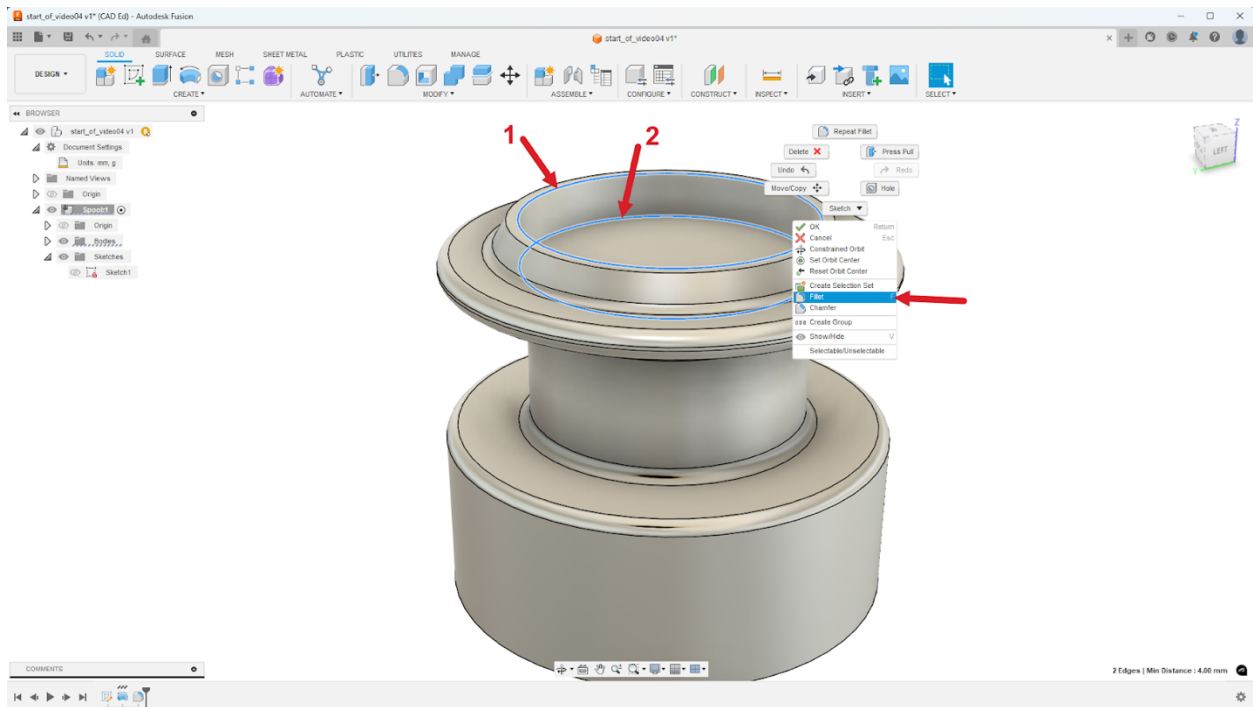
## Exhibit 2

This exhibit highlights the two edges to pre-select, and then right-click to bring up the Marking Menu. Select Fillet from the Marking Menu and enter 0.5mm for the radius.
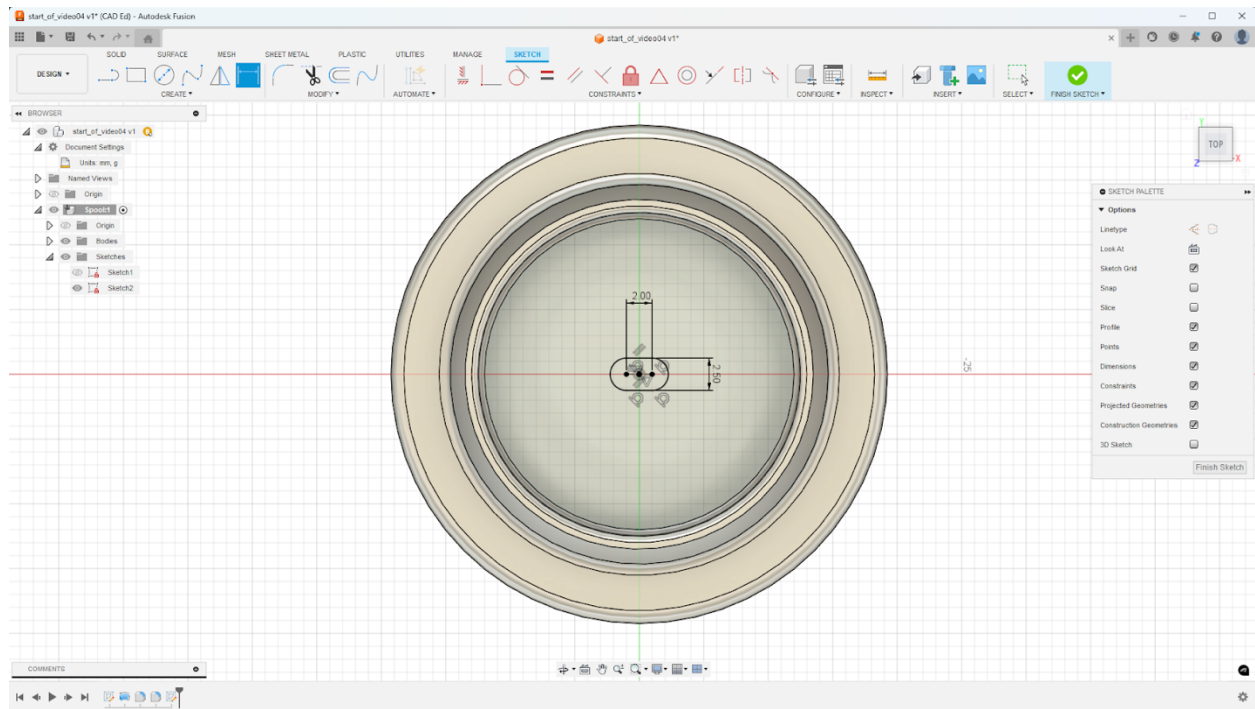
**Exhibit 3**

This exhibit shows the Center Slot at the 0,0 point with a length of 2mm and a width of 2.5mm.
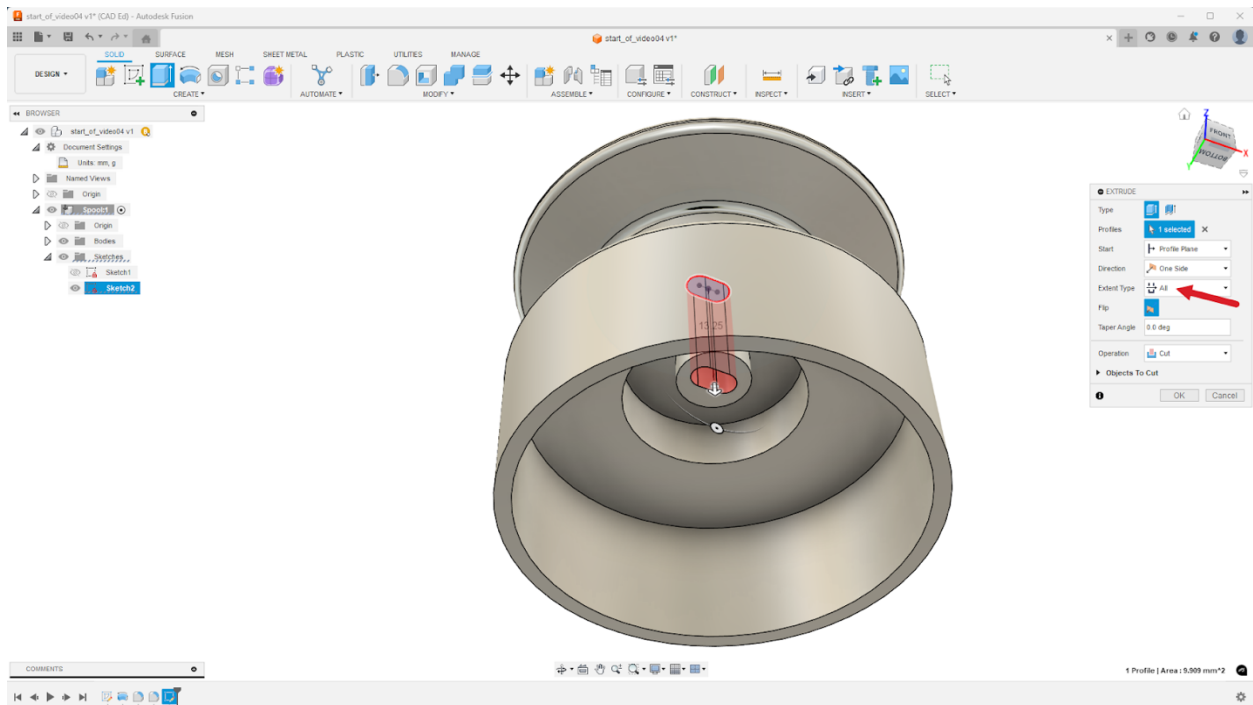
**Exhibit 4**

This exhibit shows the slot profile being extruded through the Spool. Notice the Extent Type has been set to All to make sure the profile cuts all the way through the body.
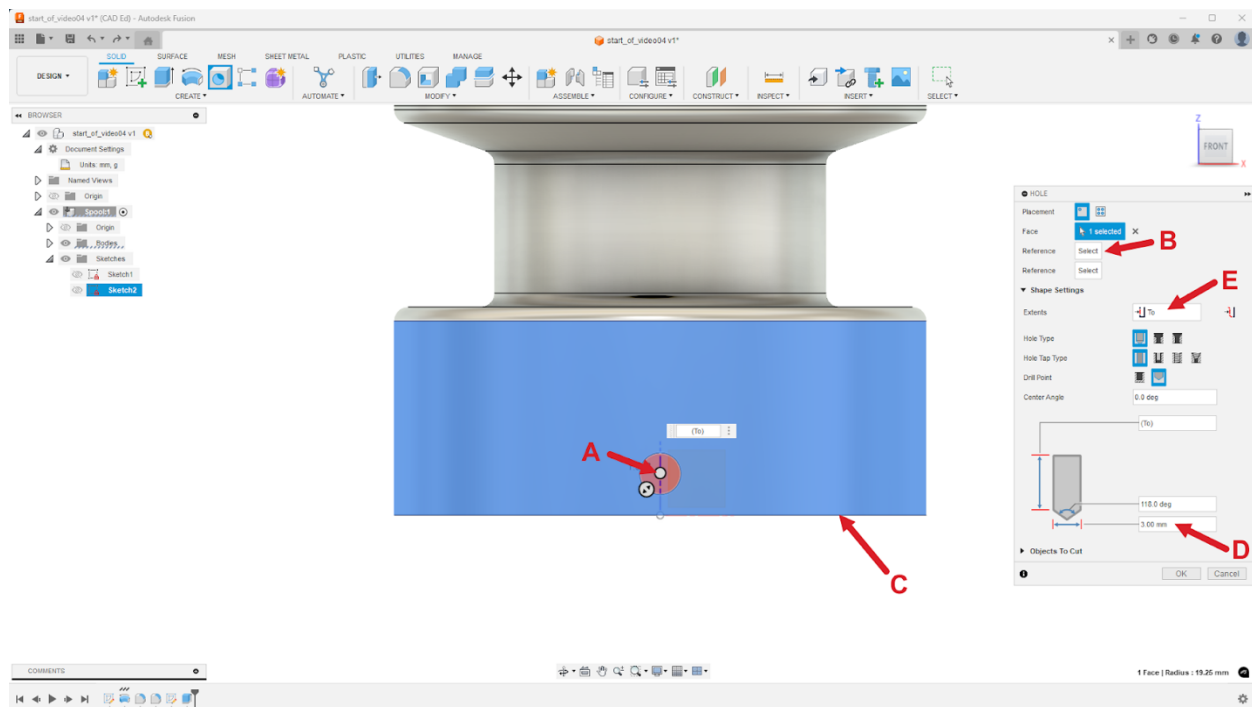
## Exhibit 5

This exhibit shows adding a hole to the cylindrical face of the spool.  Using the Hole command, click somewhere close to location A in the exhibit. Then, click Select next to Reference (B) and click on the bottom edge (C) of the cylinder and enter 3mm for the distance.  Enter 3mm for the diameter (D) of the hole. Lastly, change the Extents setting to the To option (E) and select the inside cylindrical surface of the spool to define how far the hole should go.
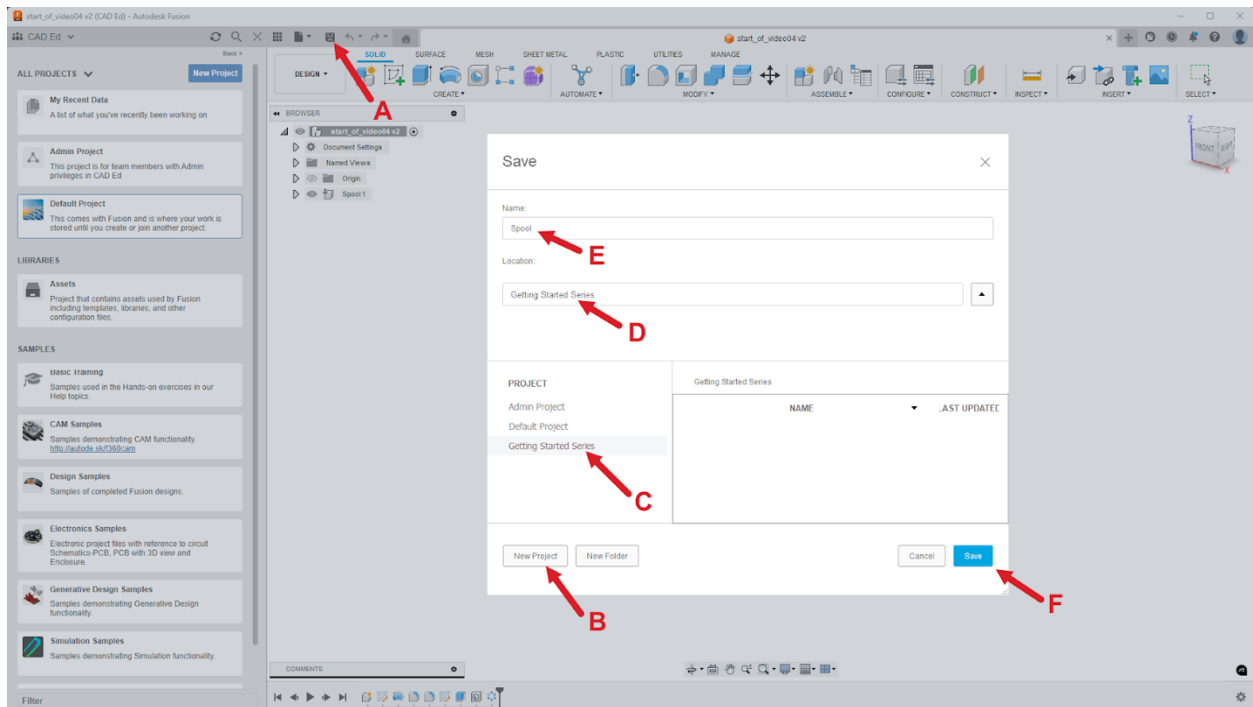
**Exhibit 6**

This exhibit shows the creation of a project and the saving of the Spool component.  First, click on the Save icon (A) in the toolbar across the top. In the dialog that appears, click on New Project (B) and enter Getting Started Series and press Enter (C). Make sure the Location is pointing to the new project (D) and then enter Spool for the Name (E). Lastly, press the Save button in the lower-right of the dialog (F). This process created a new project and then saved the Spool component into that new project.