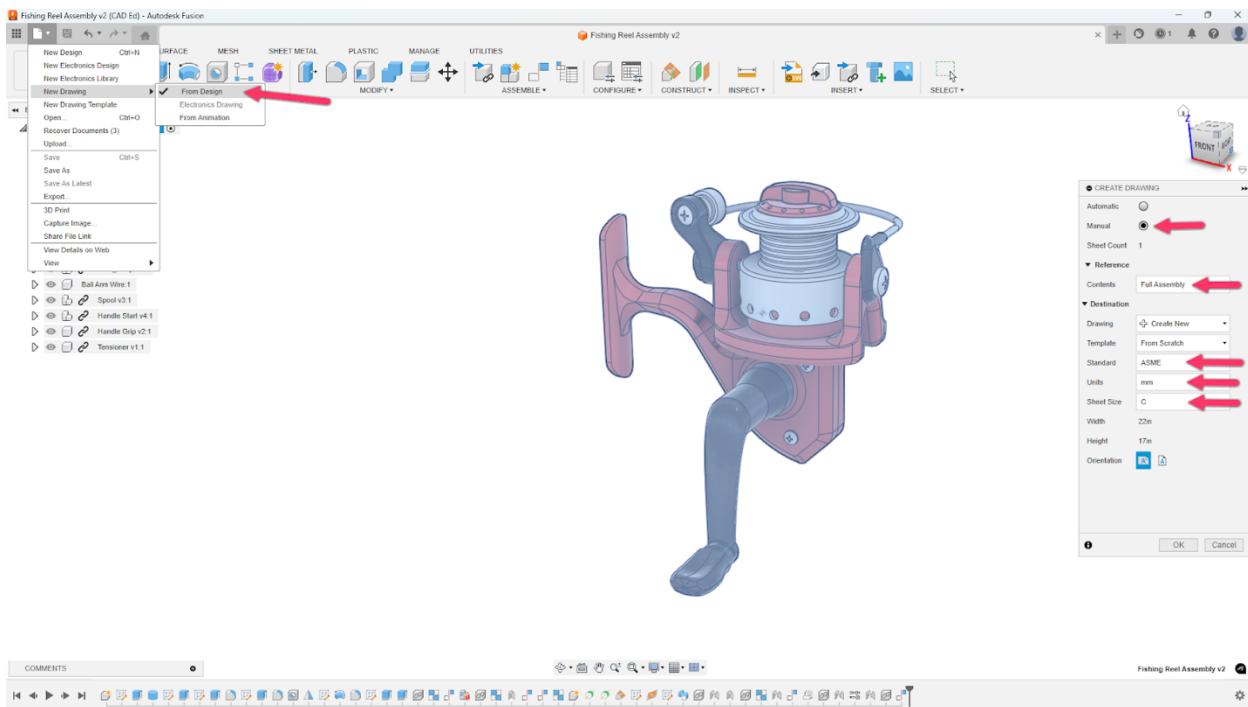


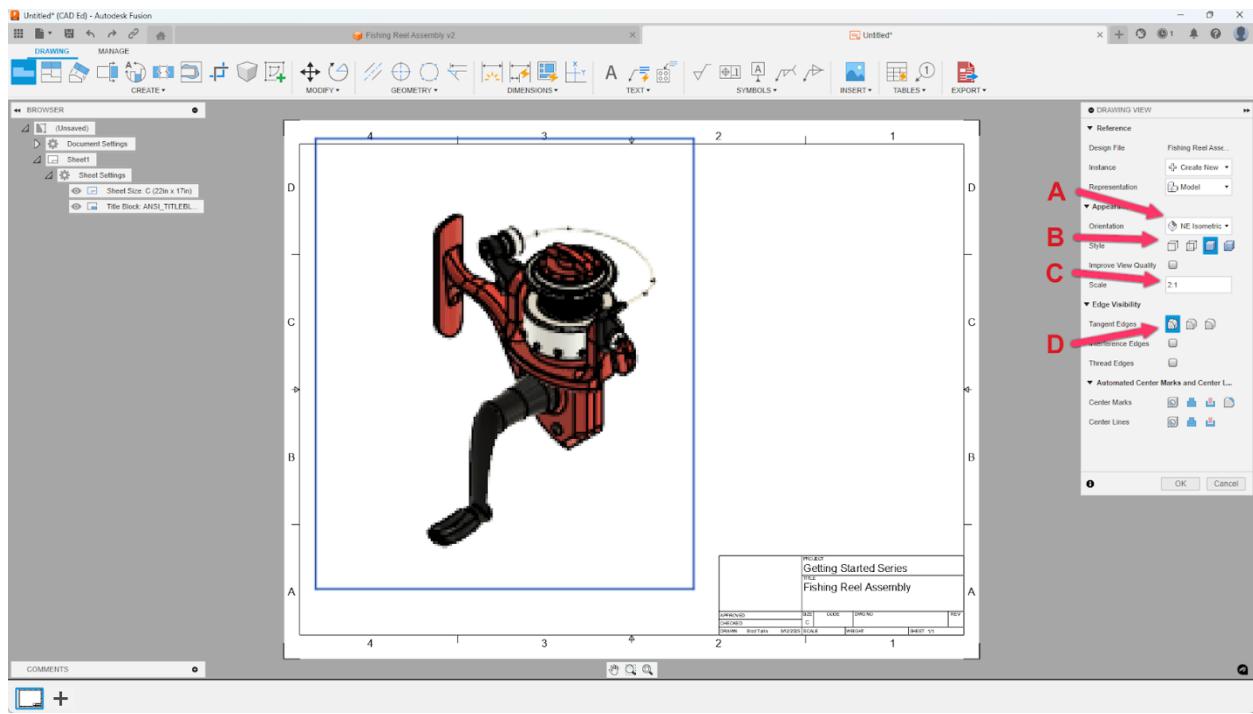
# Autodesk Fusion Getting Started tutorial series

## Video 14 - Drawings



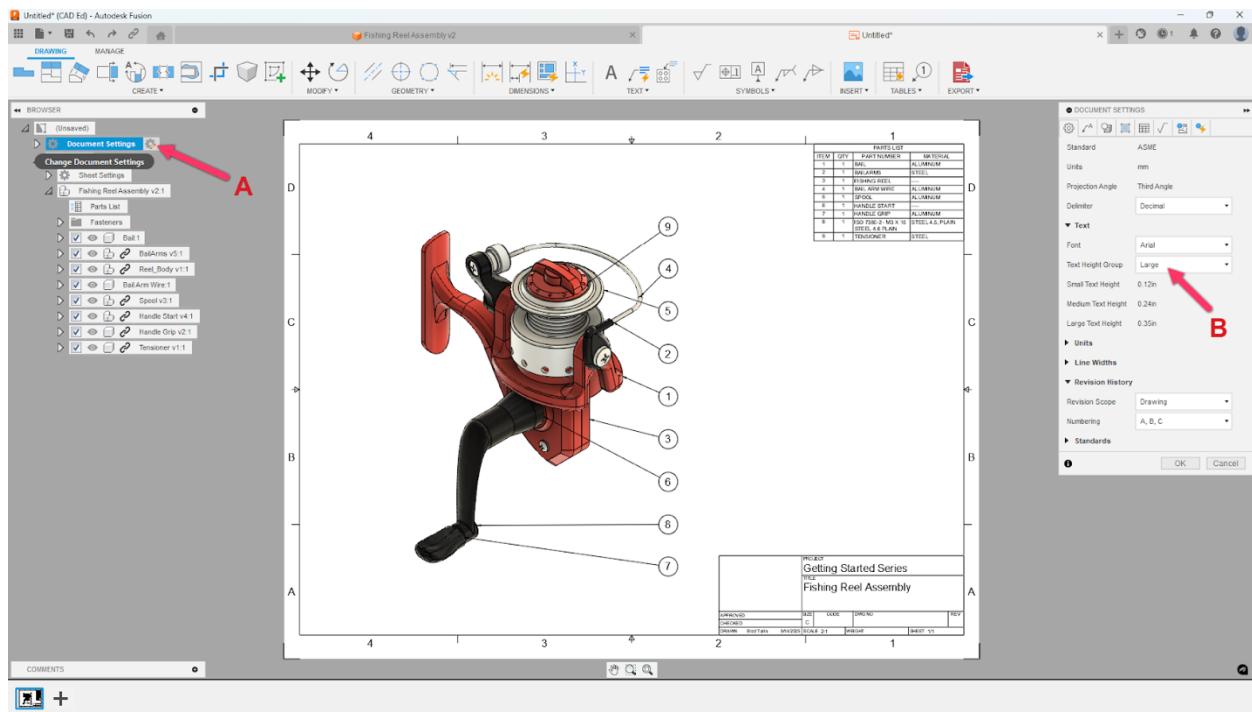
### Exhibit 1

In this exhibit, we have created a New Drawing From Design from the File Menu. It brings up the Create Drawing Dialog. Keep the Manual setting checked, then change the Standard to ASME. Make sure the Units are set to mm and change the Sheet Size to C. Then, press OK.



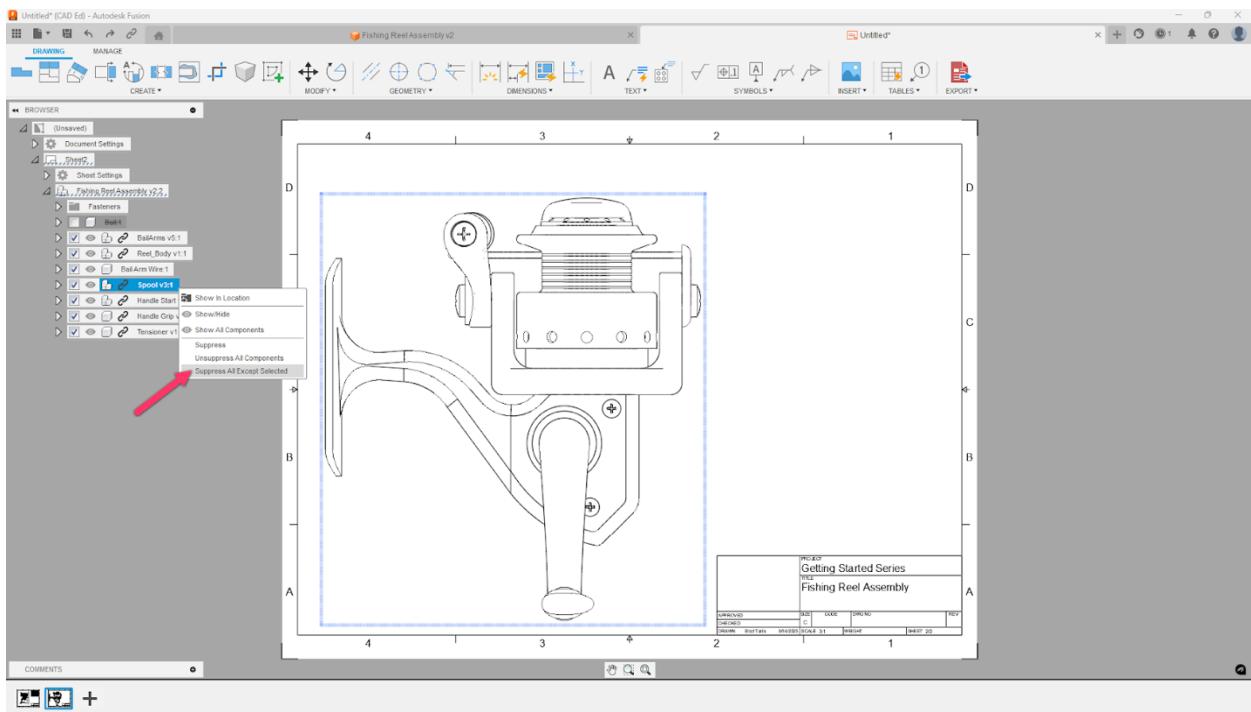
## Exhibit 2

In this exhibit, we have set the Orientation to NE Isometric (A) and set the Style to Shaded (B). Change the Scale to 2:1 (C) and set the Tangent Edges to Full Length (D). Then click the mouse in the left area of the screen to place the isometric view. Press OK to accept all of the settings.



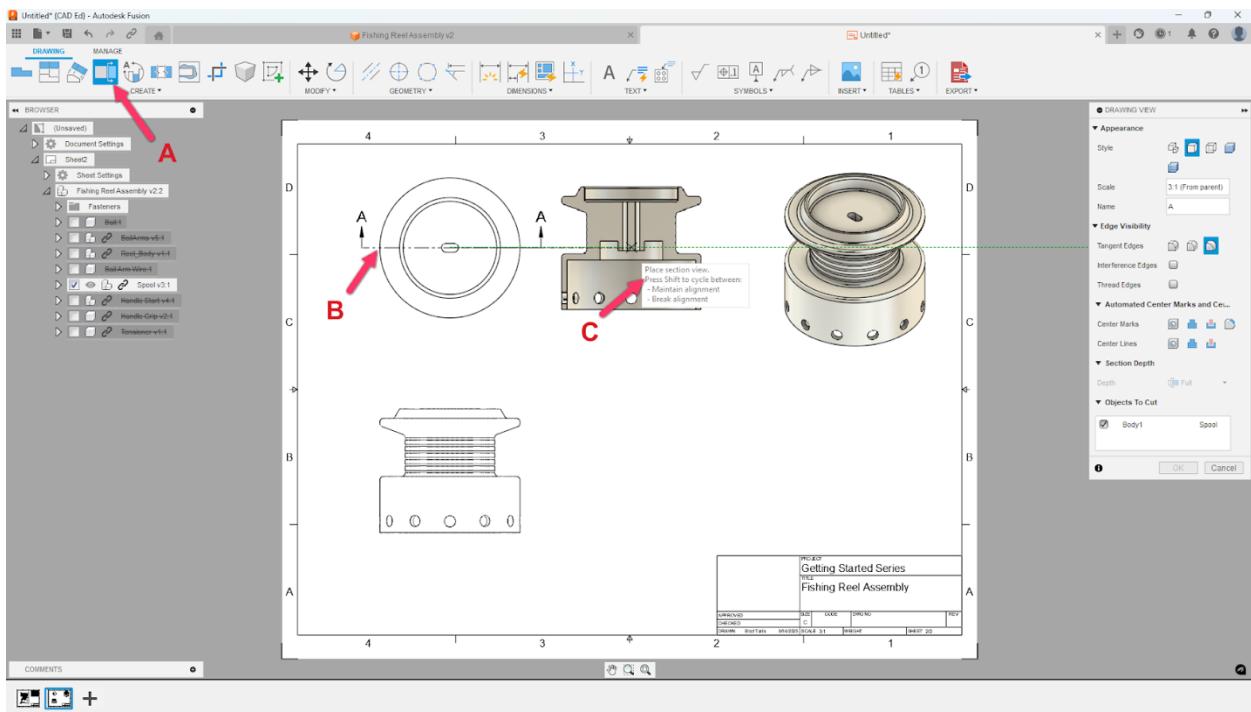
### Exhibit 3

In this exhibit, we have placed the Parts List table in the upper-right corner, but the text is a bit small. By clicking on the Change Document Settings icon (A), we can change the Text Height Group from Medium to Large (B). Then press OK and the table will update.



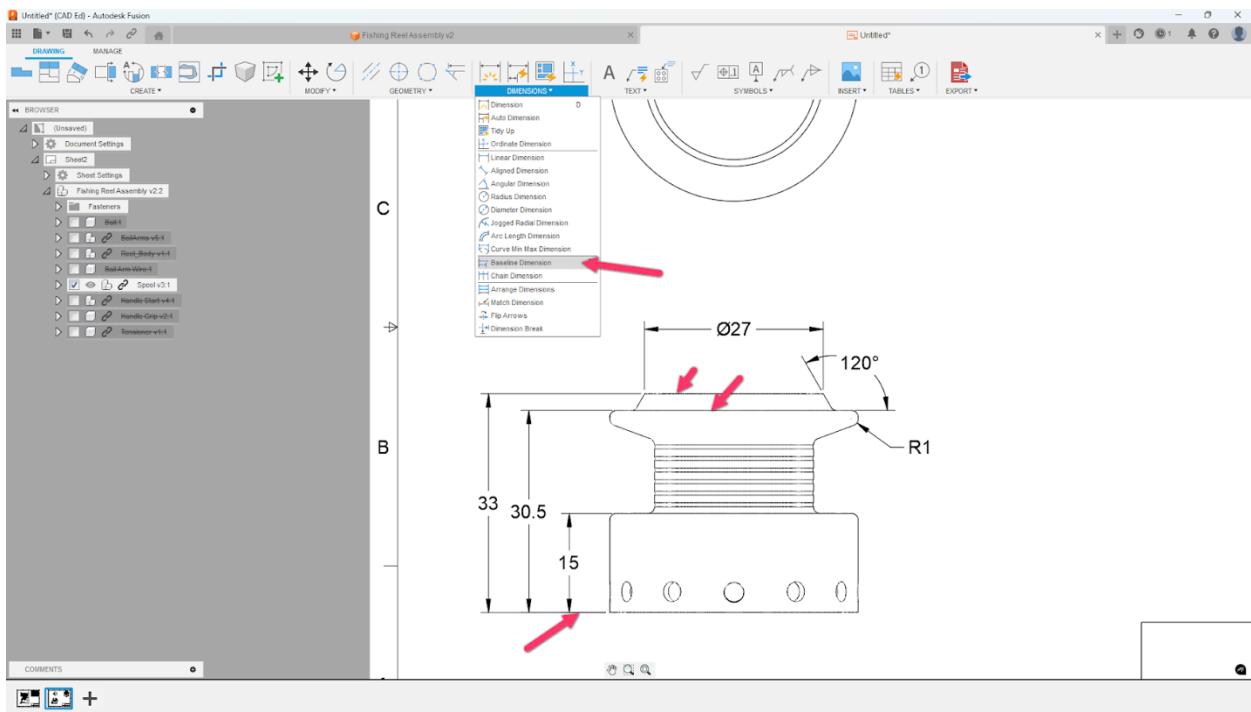
#### Exhibit 4

In this exhibit, we have created a new sheet, then placed a base view of the whole assembly. Right-click on Spool on the Browser and select Suppress All Except Selected to turn off all of the other components except the spool.



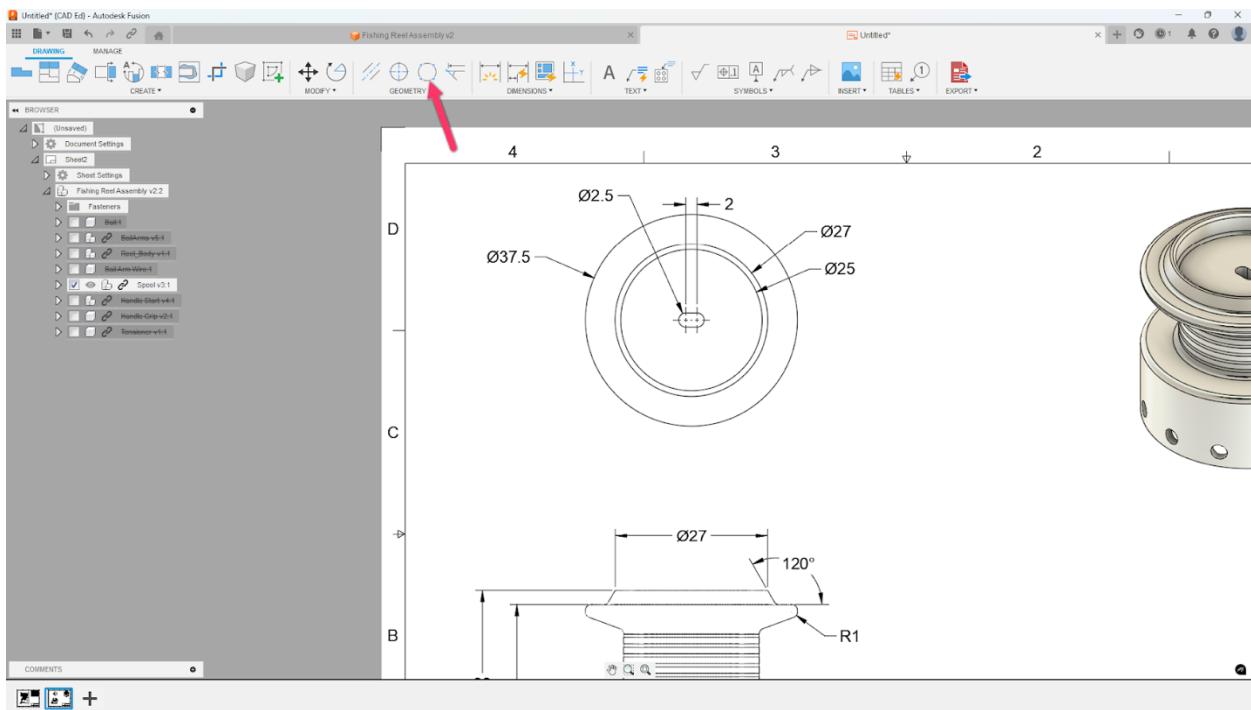
## Exhibit 5

In this exhibit, we have clicked on the Section View command from the main menu (A). Then, select the Top View as the parent view and hover over point B and move your mouse to the left and click to place the start of the section line. Move to the right past the edge of the spool and click to place the end of the horizontal section line. Press the Shift Key once to toggle the alignment (C) of the section view.



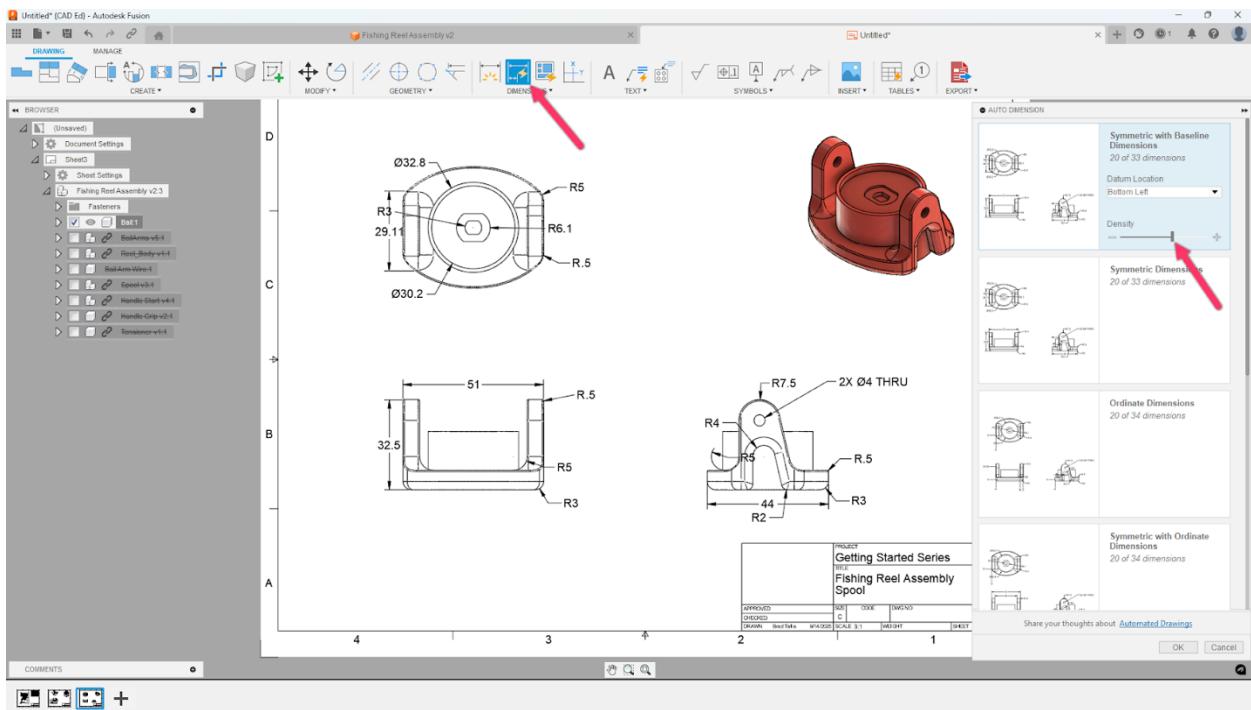
## Exhibit 6

In this exhibit, we are using the Baseline Dimension command from the Dimension menu to create Baseline dimensions. Click on the bottom of the 15mm dimension extension line, then click the two horizontal lines of the spool to create the 30.5 and the 33mm dimensions.



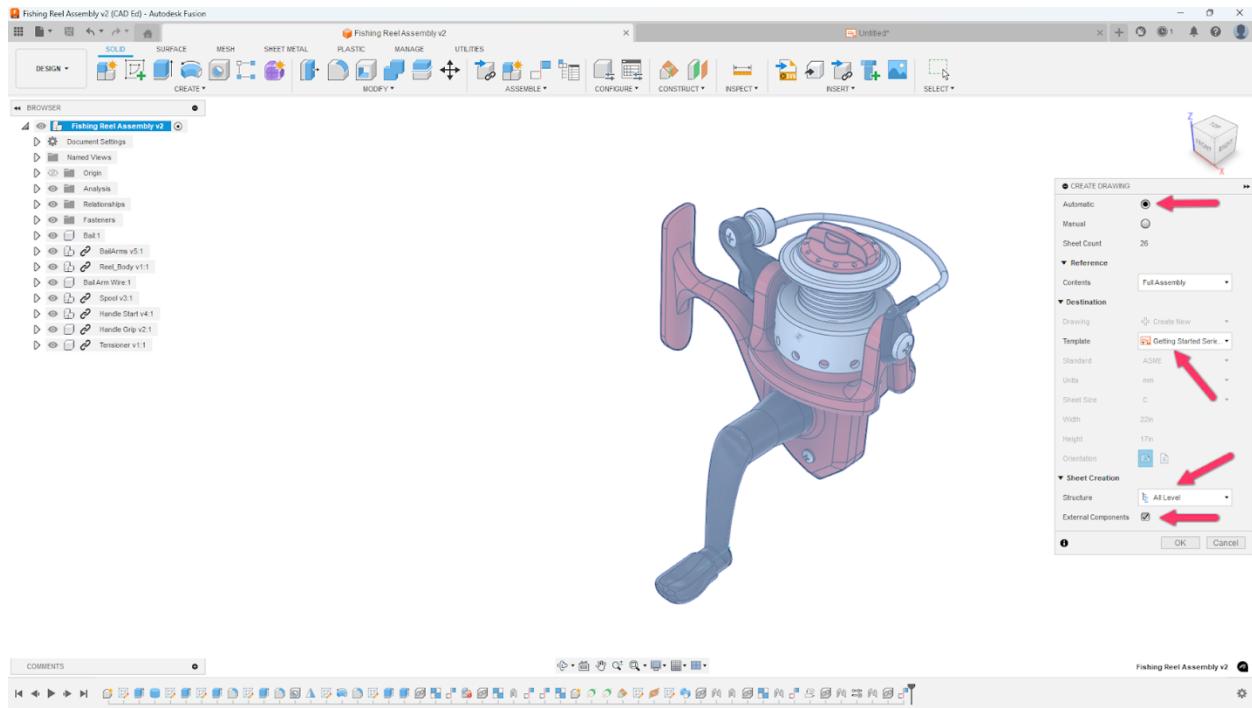
### Exhibit 7

In this exhibit, we have used the Center Mark Pattern command to create center marks on the slot feature. Then, using the dimension tool, dimension the length and diameter of the slot. Finally, add the diameter dimensions to the circles of the Top View.



## Exhibit 8

In this exhibit, we have run the Auto Dimension command from the Dimension menu. It brings up the Auto Dimension dialog and allows you to select which style of dimensions you want created. You can also adjust the Density slider to add or remove the number of dimensions displayed.



## Exhibit 9

[Link for Getting Started Series TB](#)

In this exhibit, we are creating a new drawing, but setting the option to Automatic. Change the Template to the Getting Started Series TB file you uploaded into the project. Change the Structure to All Level and make sure External Components is checked, then press OK.