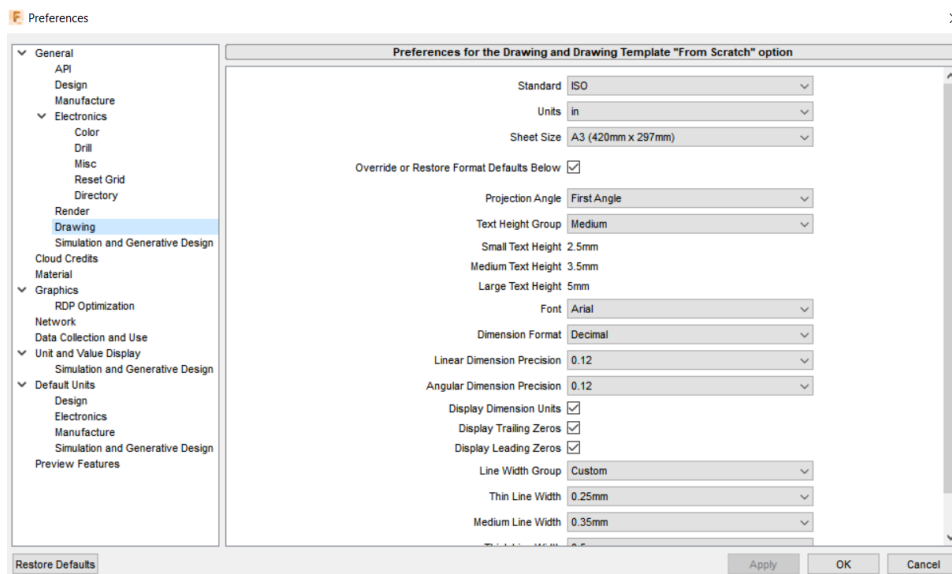


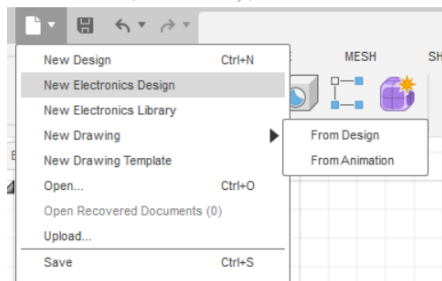
Start with a component or body in Fusion 360

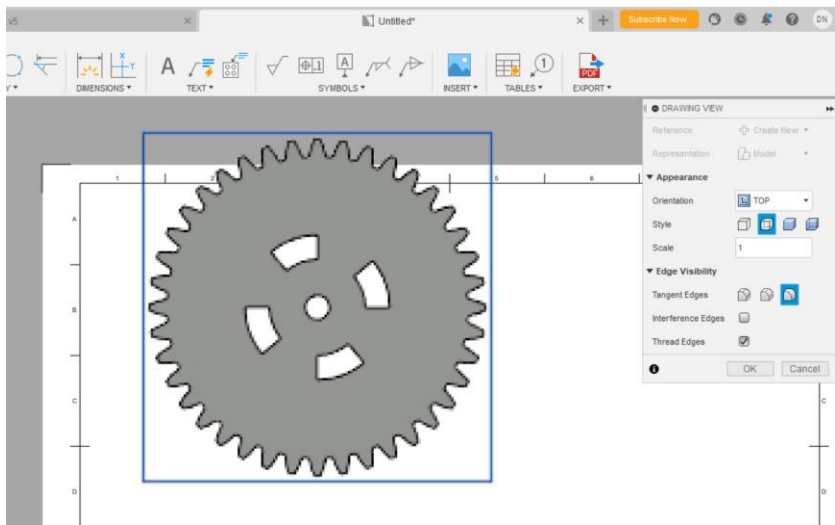
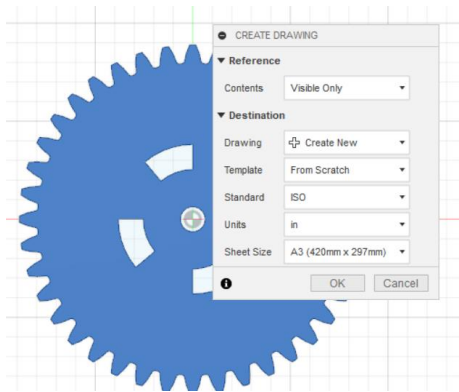
First, left click on the top right icon with your name, and select preferences. Select Drawing in the left side, and then change parameters to ISO, and set units to Inch.



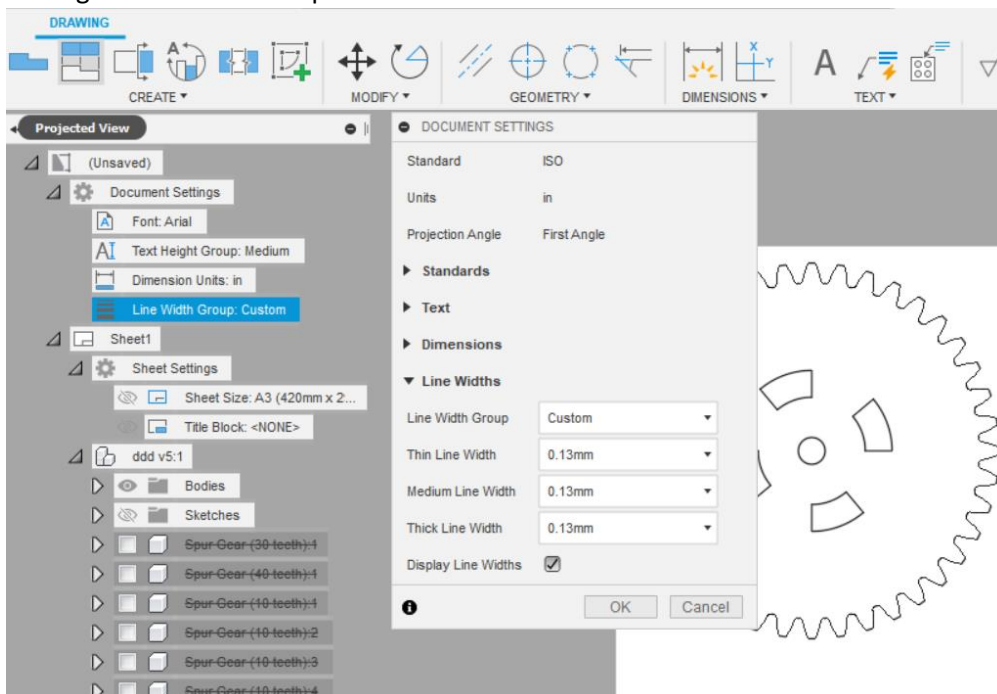
Then, arrange your parts in the 3D space with all the faces you want to cut facing the same plane. Leave the Bodies and Components you want to cut set to Visible, and all others not visible.

At top left, click File<New Drawing<From Design





From here you can delete the formatting and borders. Also, on the left side, select Document Settings<Line Width Group and edit the lines.



Now you can export your drawing as a PDF in the top right corner. When you do, make sure Line Weights are OFF/Unchecked. This should enable the Laser Cutters in the Innovation Studio to read your lines for cutting.

