Autodesk Fusion Getting Started tutorial series
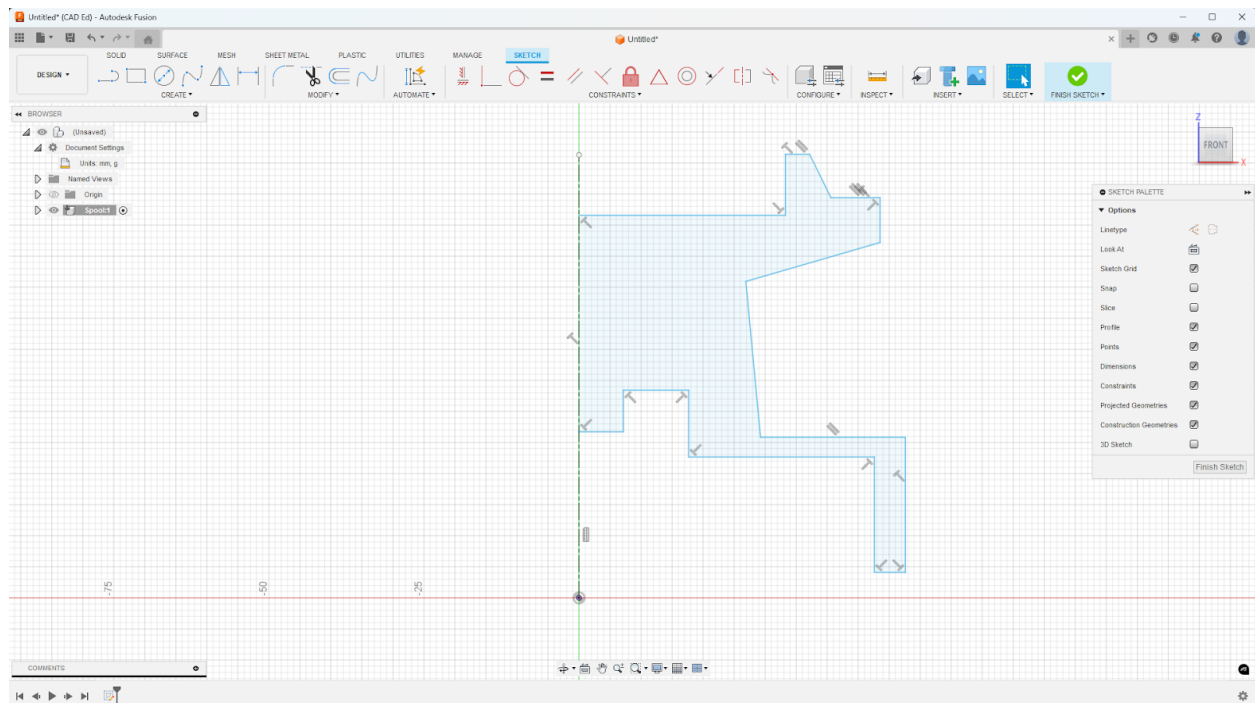
Video 3 - Creating a Sketch

Exhibit 1

In this exhibit, we have drawn a line vertically from the 0,0 point on our sketch and changed it to a centerline linetype. Then, using the line command, roughly draw the side profile of the spool. Do not worry about dimensions at this time.
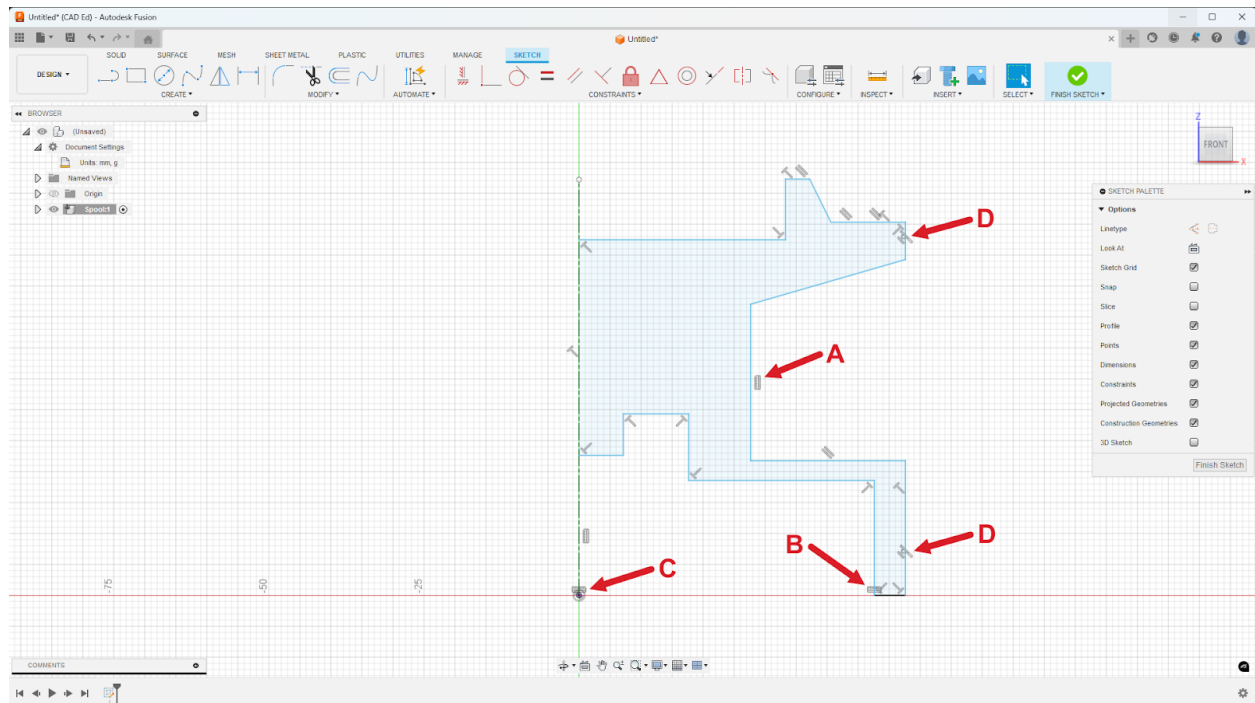
Exhibit 2

In this exhibit, we have added a vertical constraint to the line at A, and a horizontal constraint from point B to Point C. Make the two outer edges labeled D inline with each other by using a Collinear constraint.
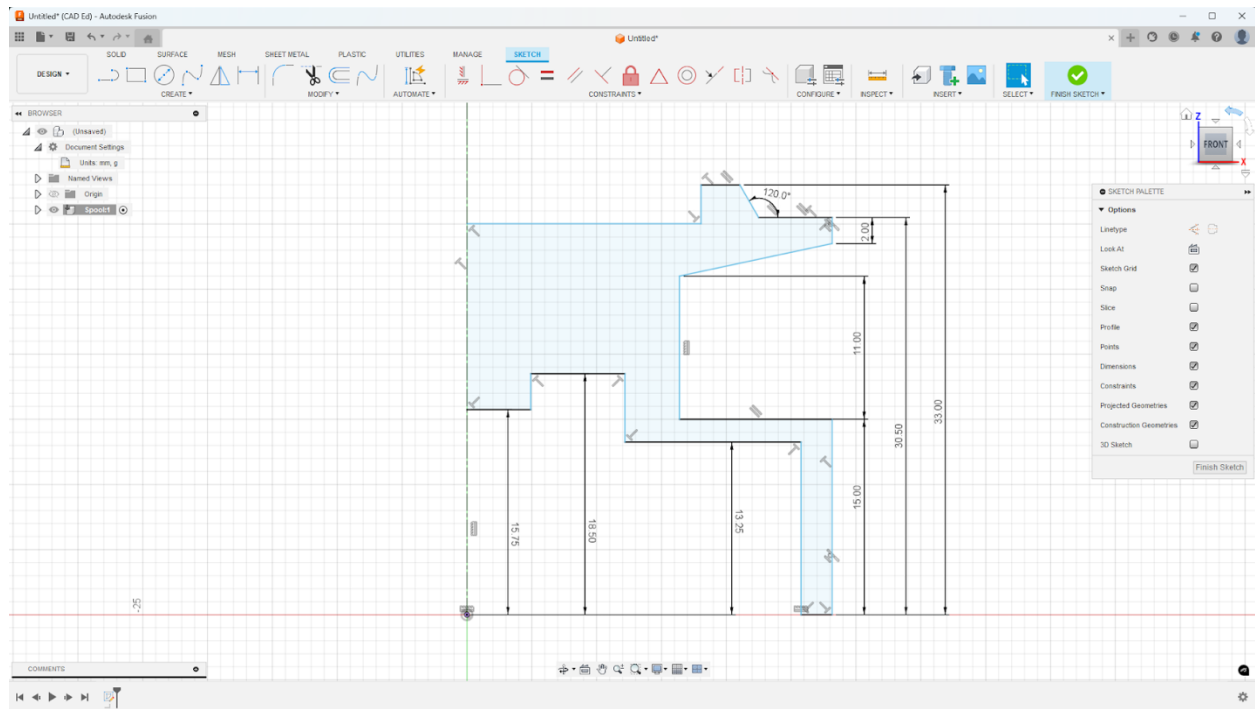
Exhibit 3

In this exhibit, we have added all the vertical dimensions and the 120° angle dimension.
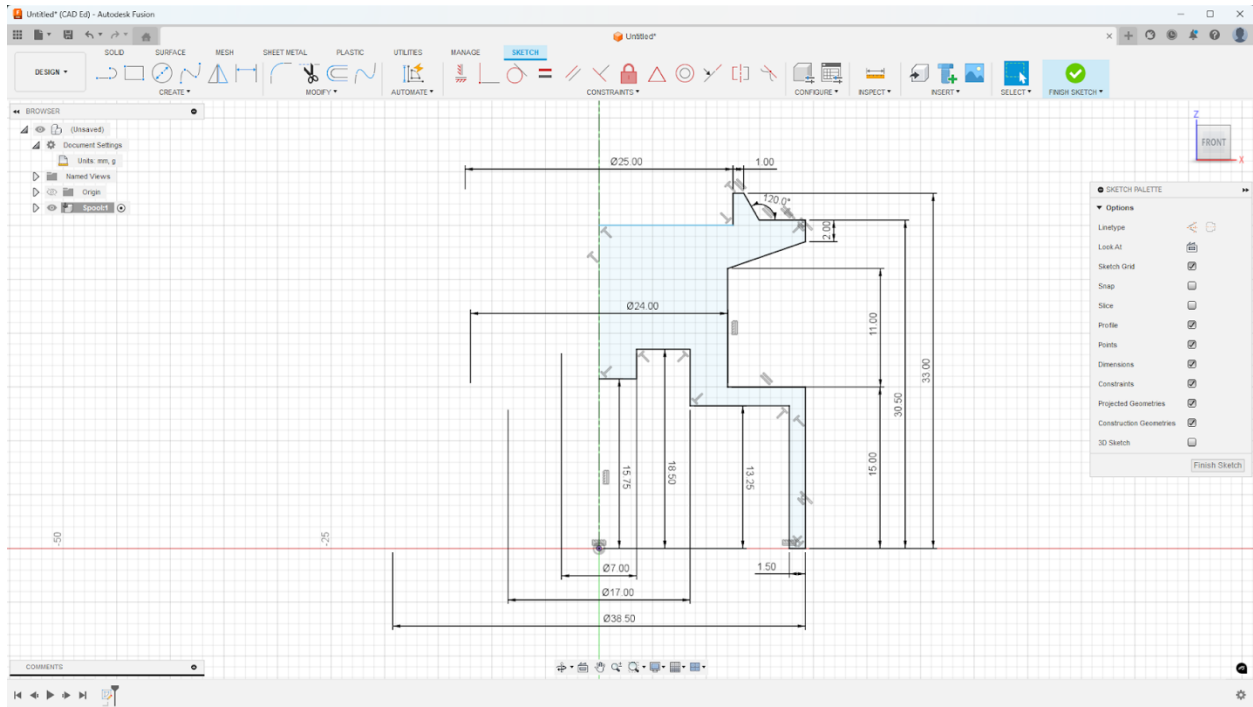
Exhibit 4

In this exhibit, we have added the horizontal dimensions. By clicking the centerline first, then a vertical edge, it automatically creates a diameter dimension, as indicated by the diameter symbol in front of the dimension.