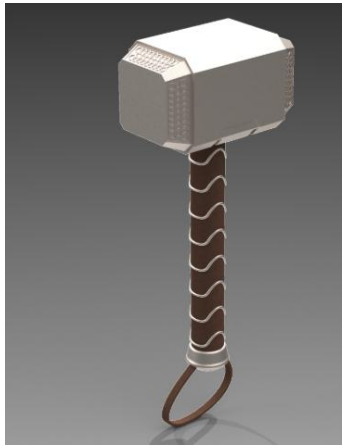
Tutorial creation inventor

I chose to model Thor's hammer because it is an emblem of the cartoons from our childhood and I am going to show you how to model it in 3D using Inventor.



Tutorial creation inventor



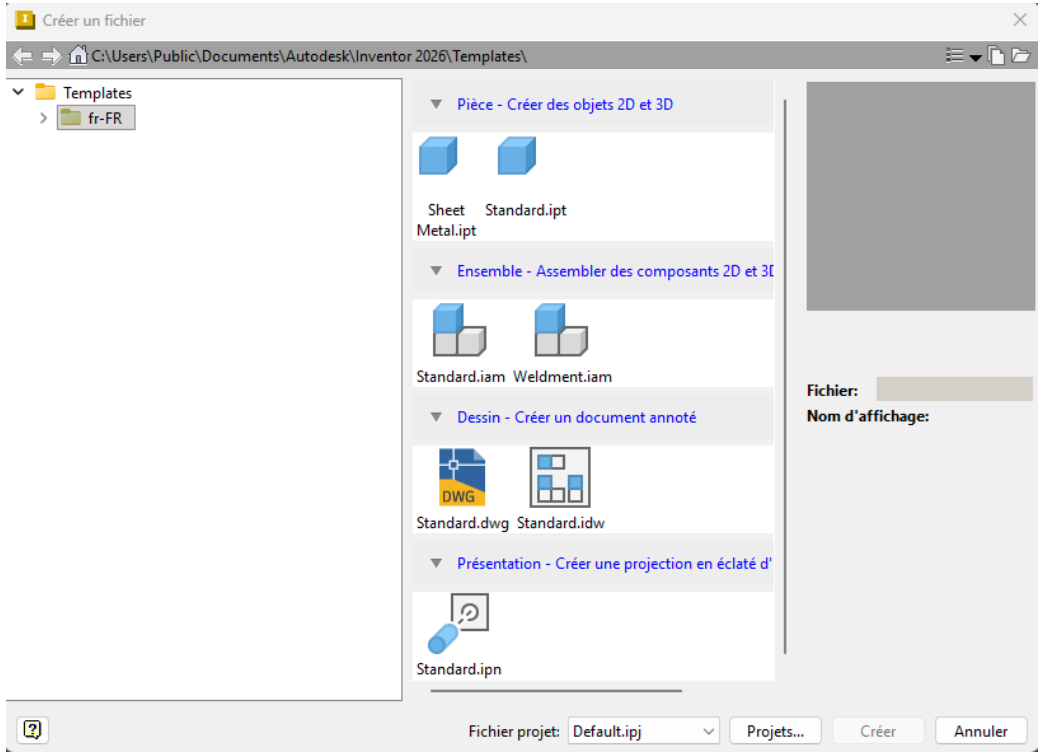
Step 1:

To begin, you need to open Inventor

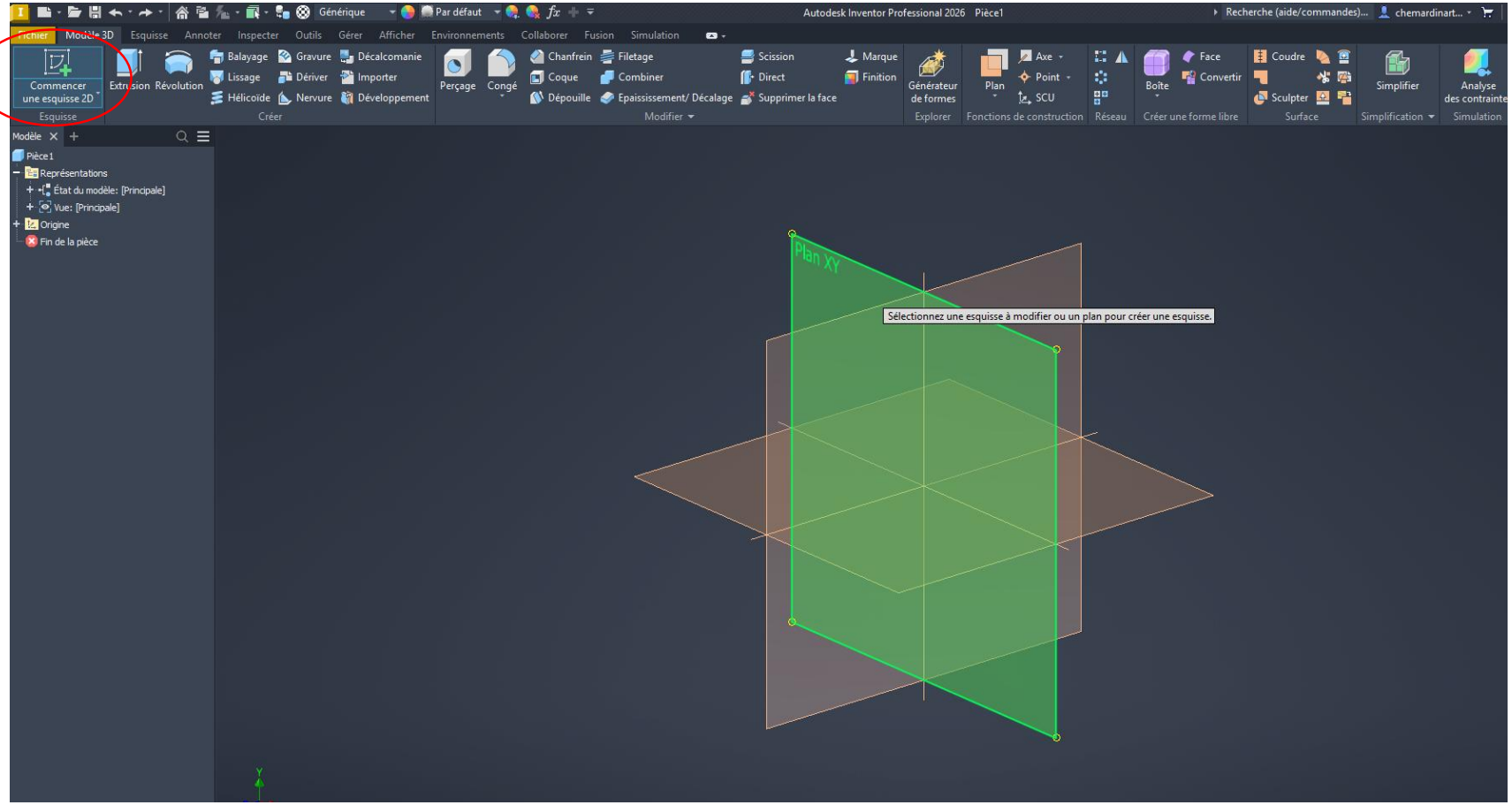


Step 2:

Create a new file standars.ipt

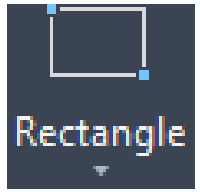


Step 3:

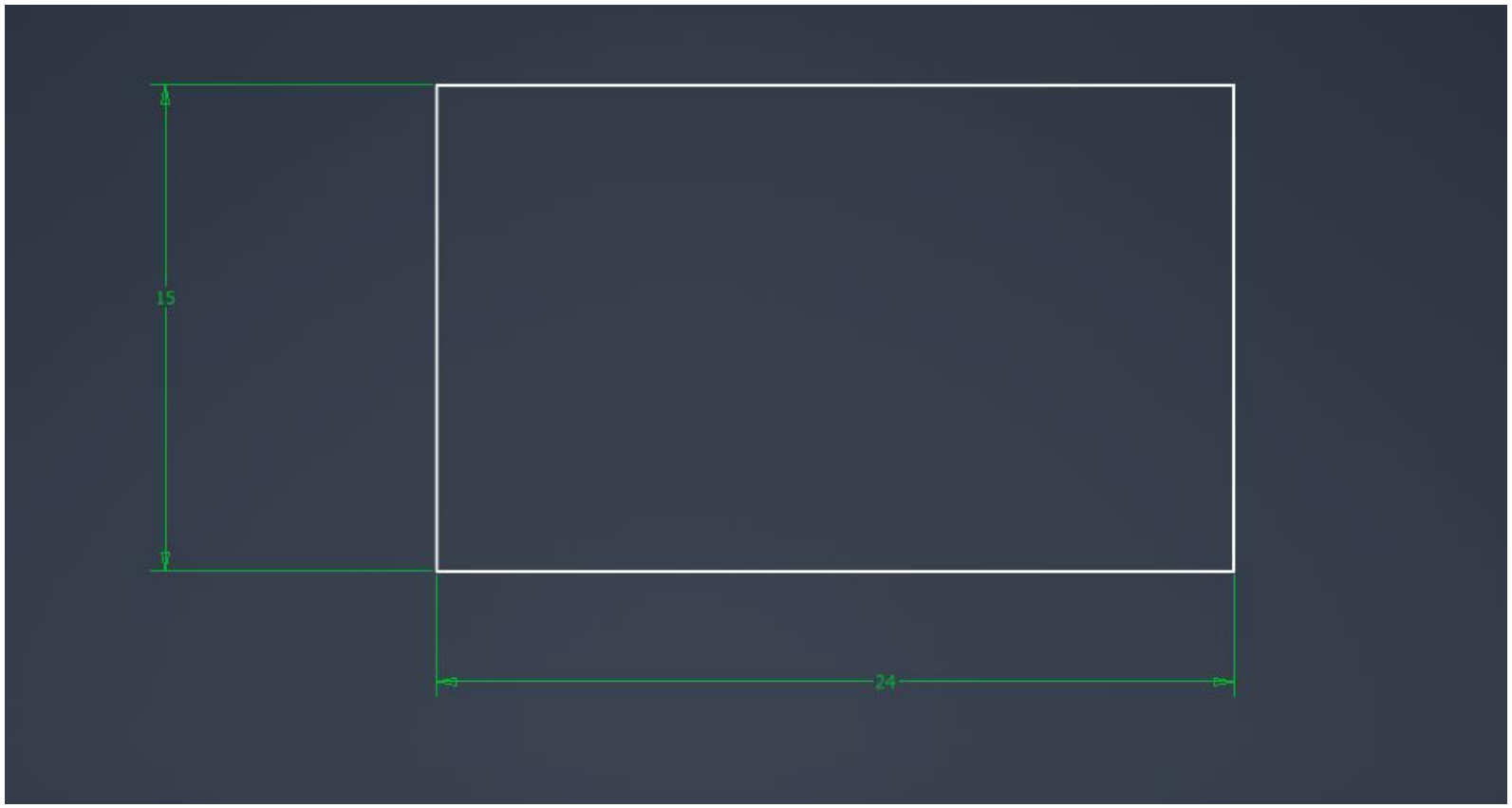


Start a 2D sketch and select the XY plane

Step 4:



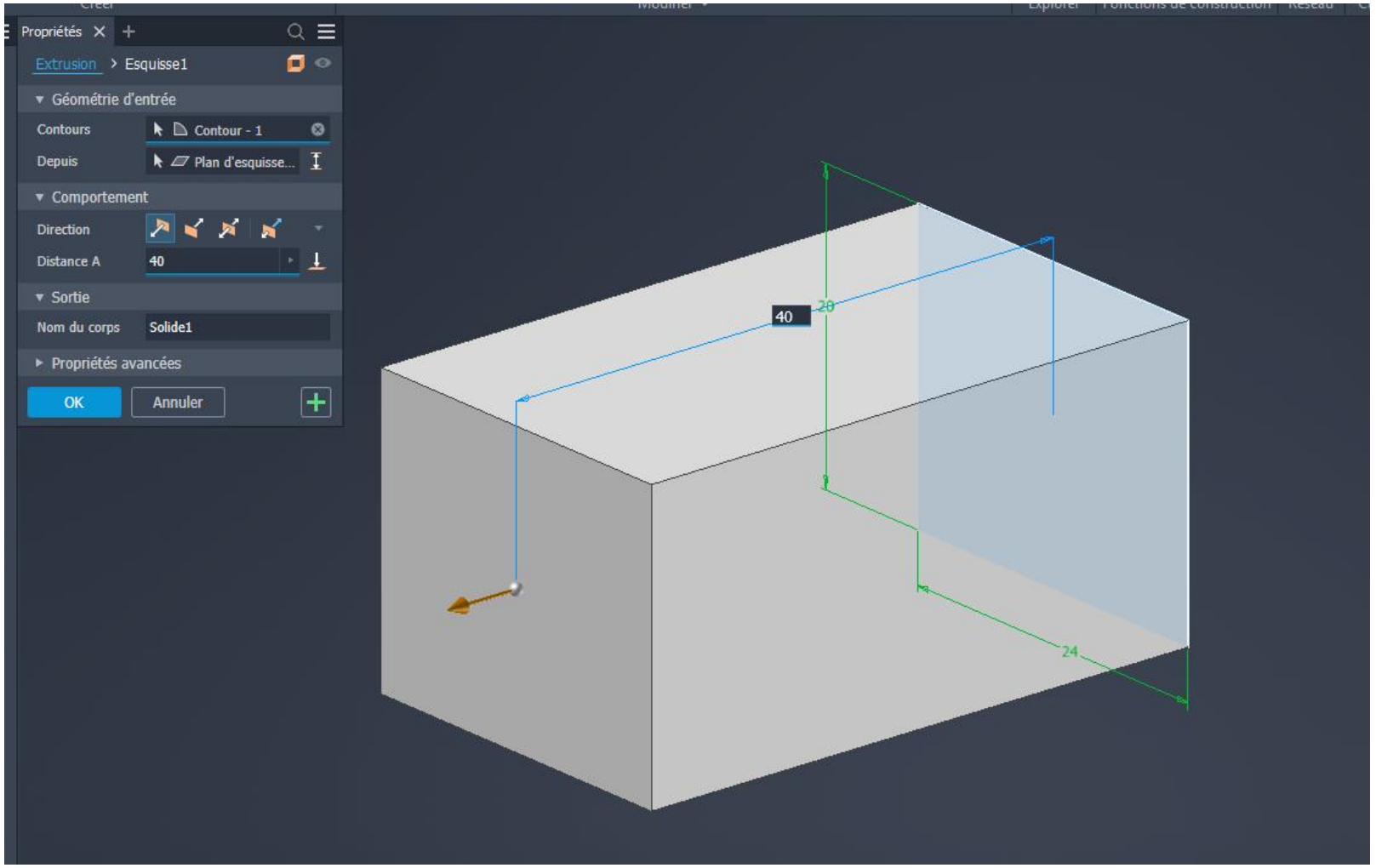
Create a rectangle with dimensions: 24 in length and 15 in height



Step 5:

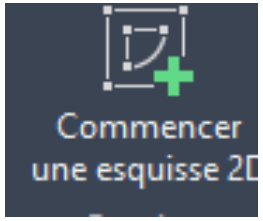


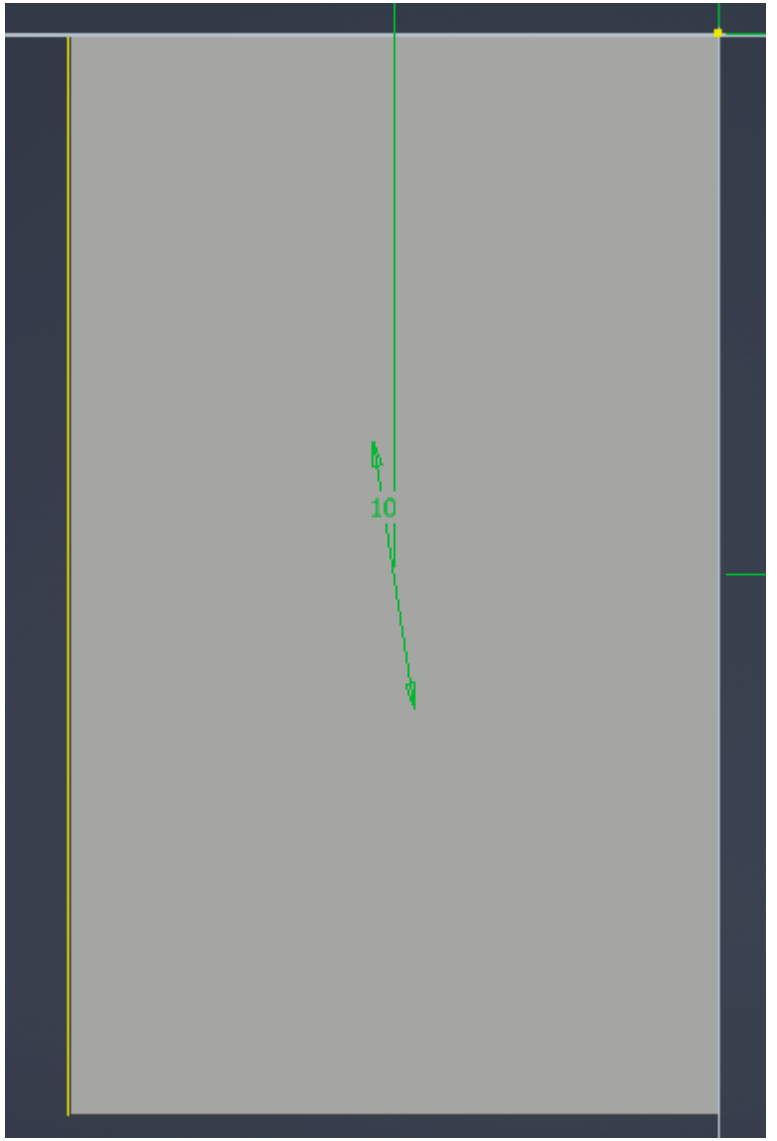
Finish the sketch and extrude by 40



Step 6:

Create a new 2D sketch
and select the face

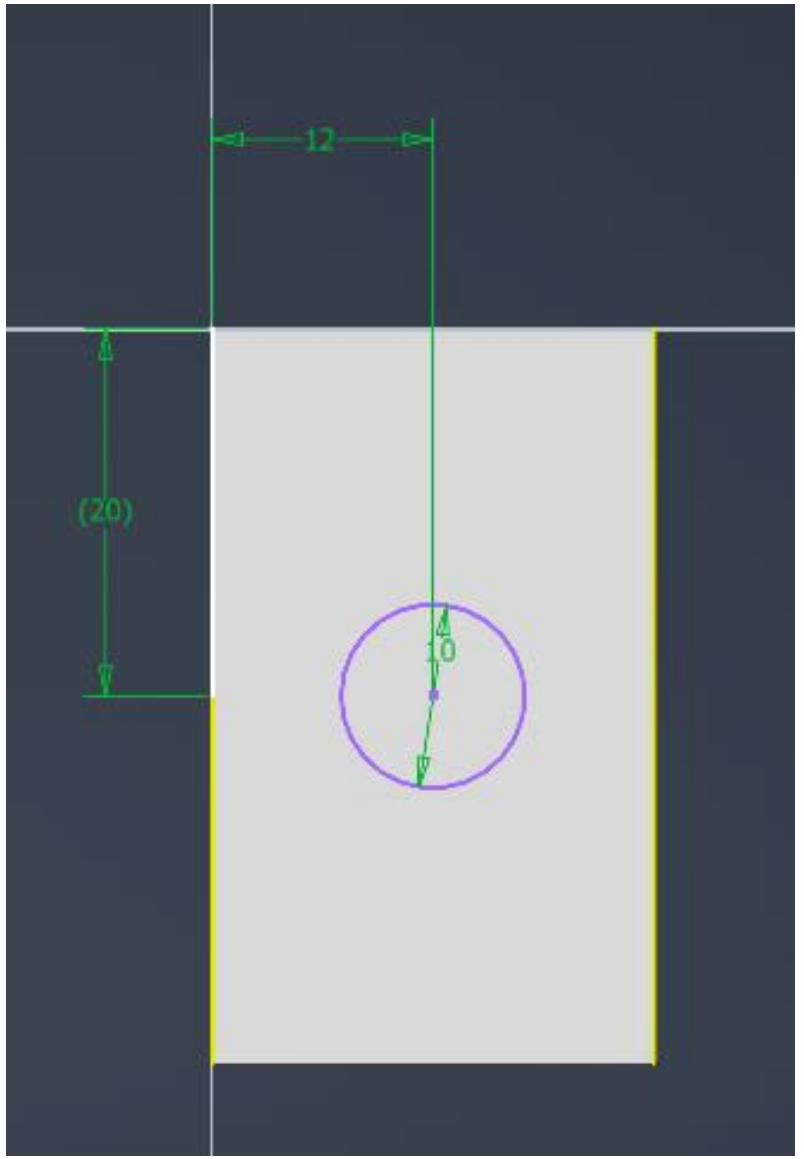




Step 7:



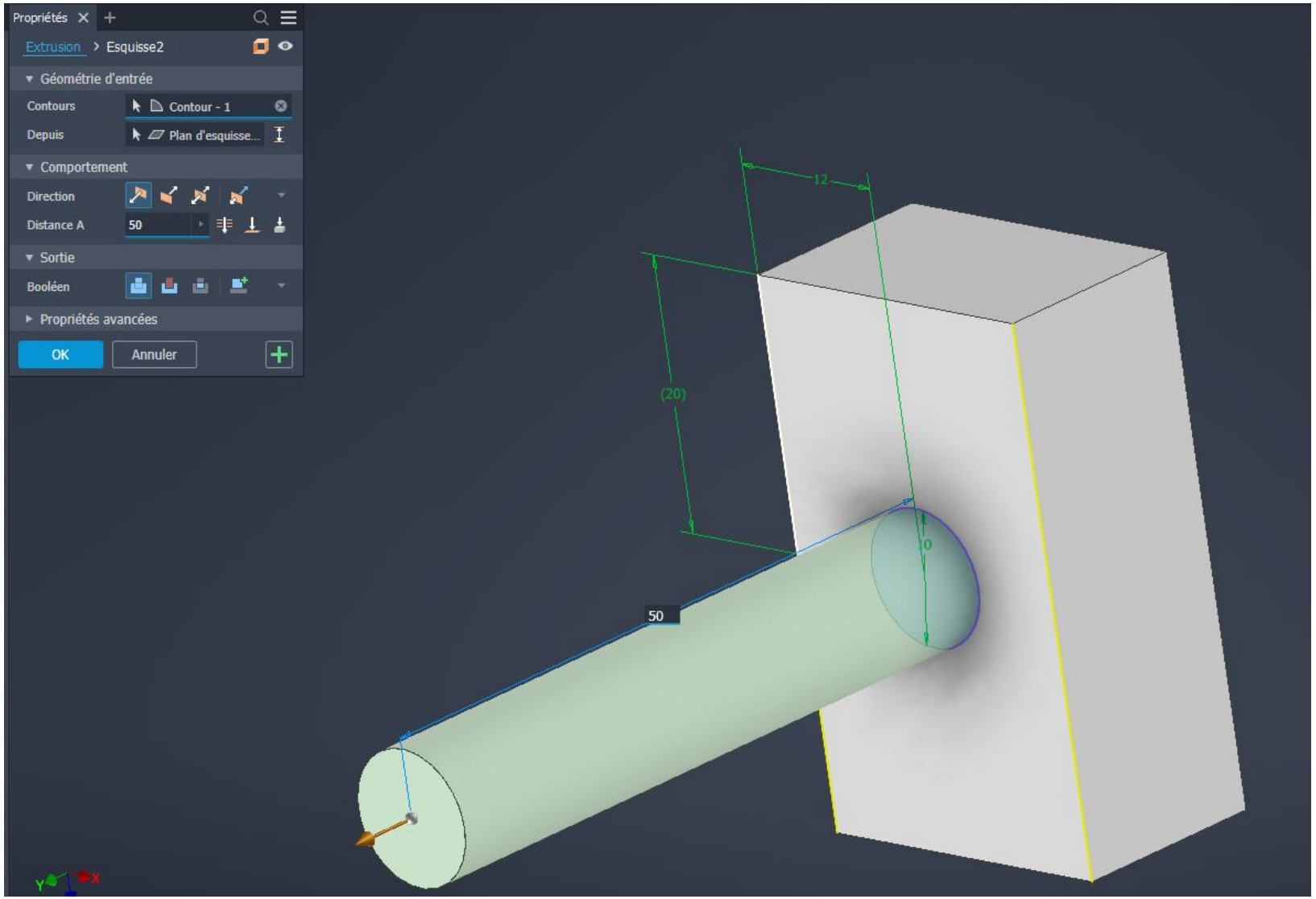
Create a circle at the center of the selected face



Step 8:



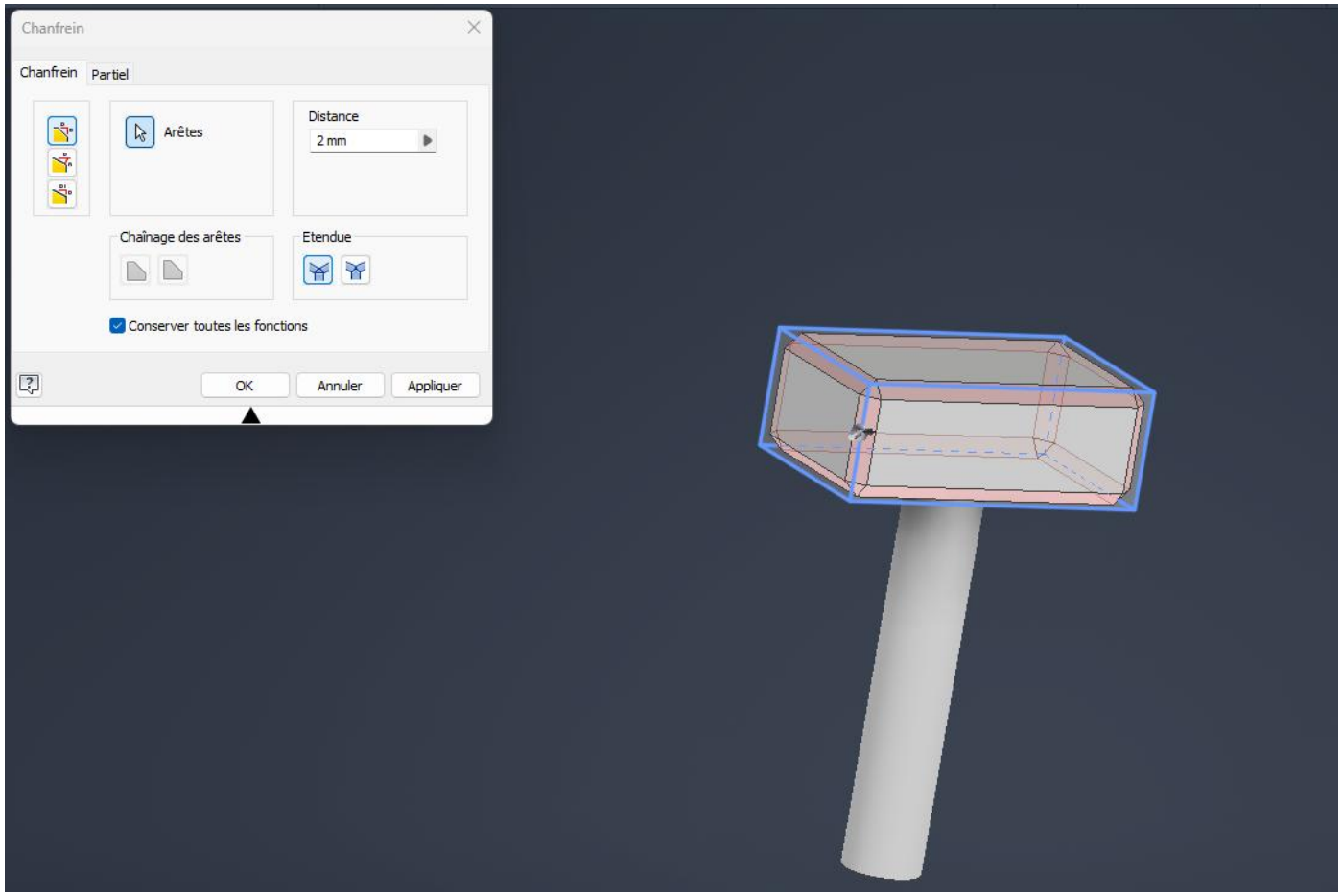
Create a extrusion
in the circle of 50



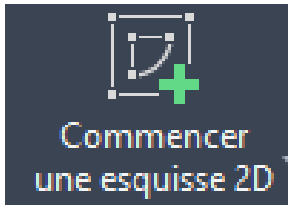
Step 9:

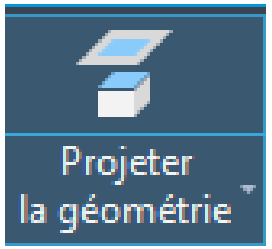


select the edges and make a chamfer

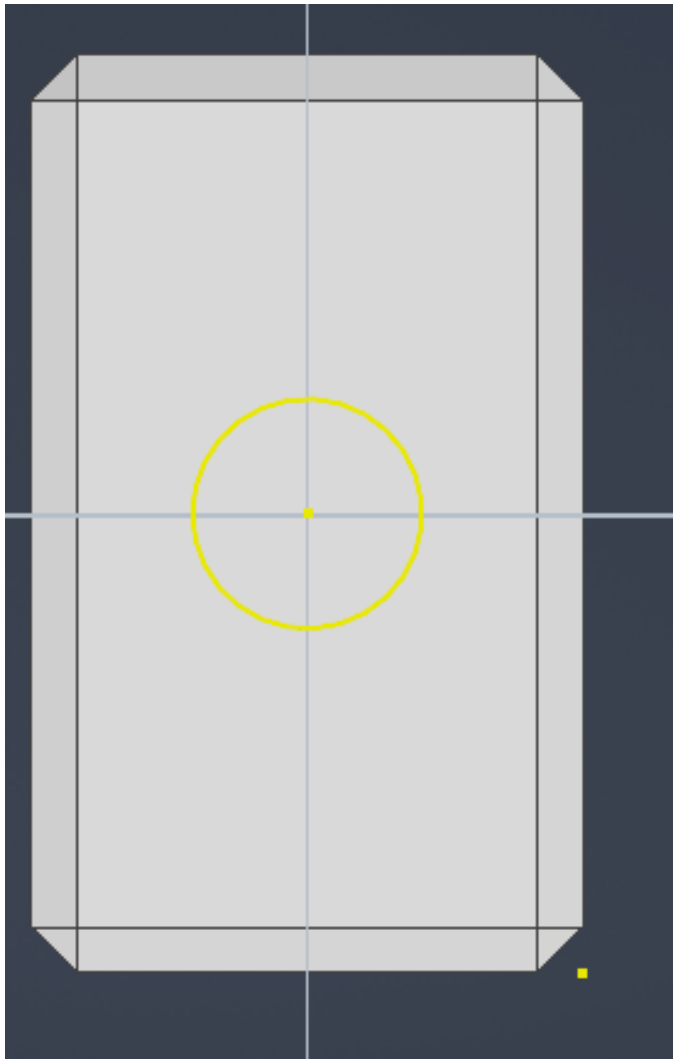


Step 10:

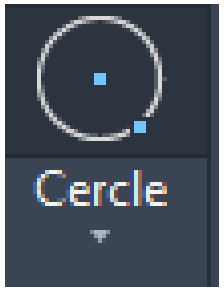




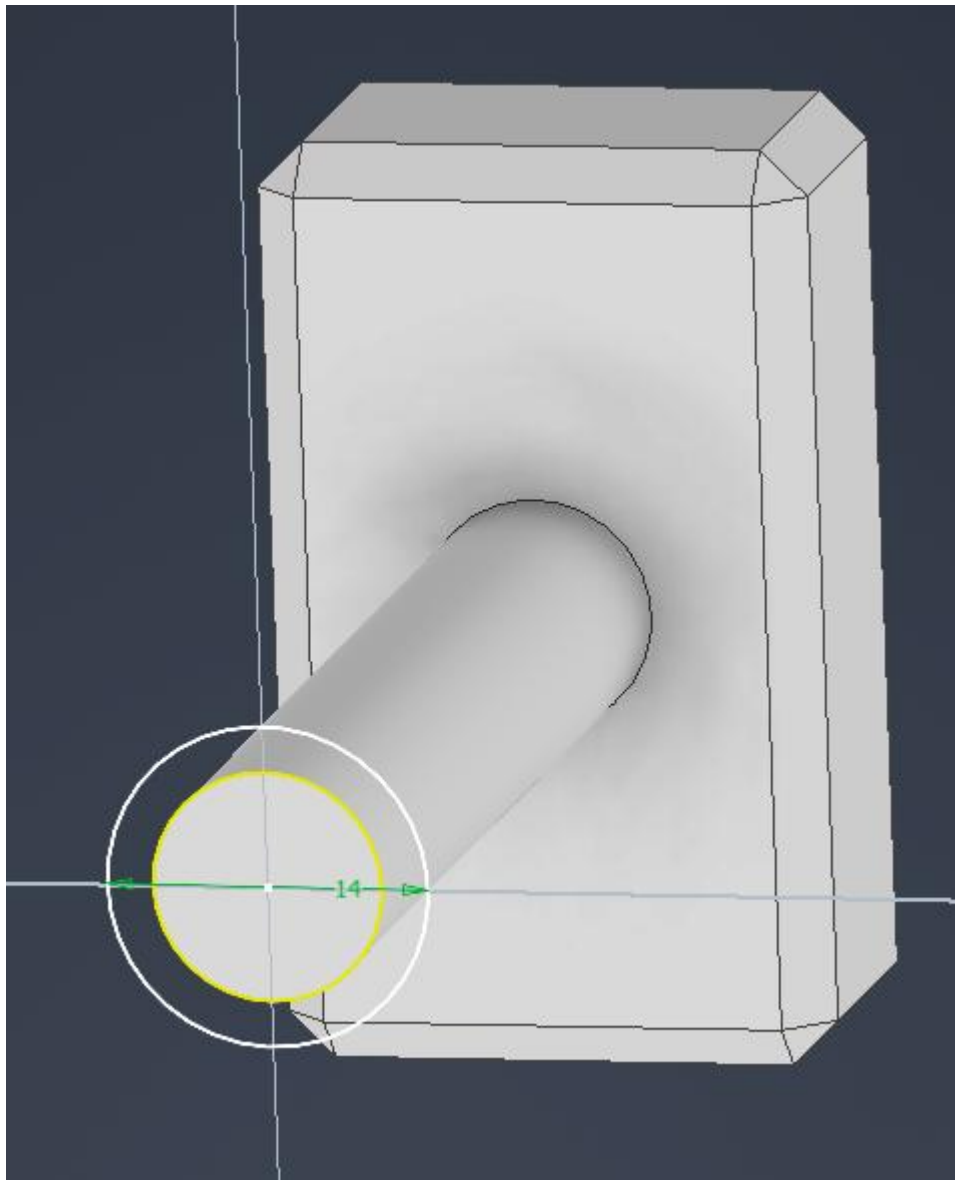
start a new sketch at the end of the tube and project the geometry onto it



Step 11:



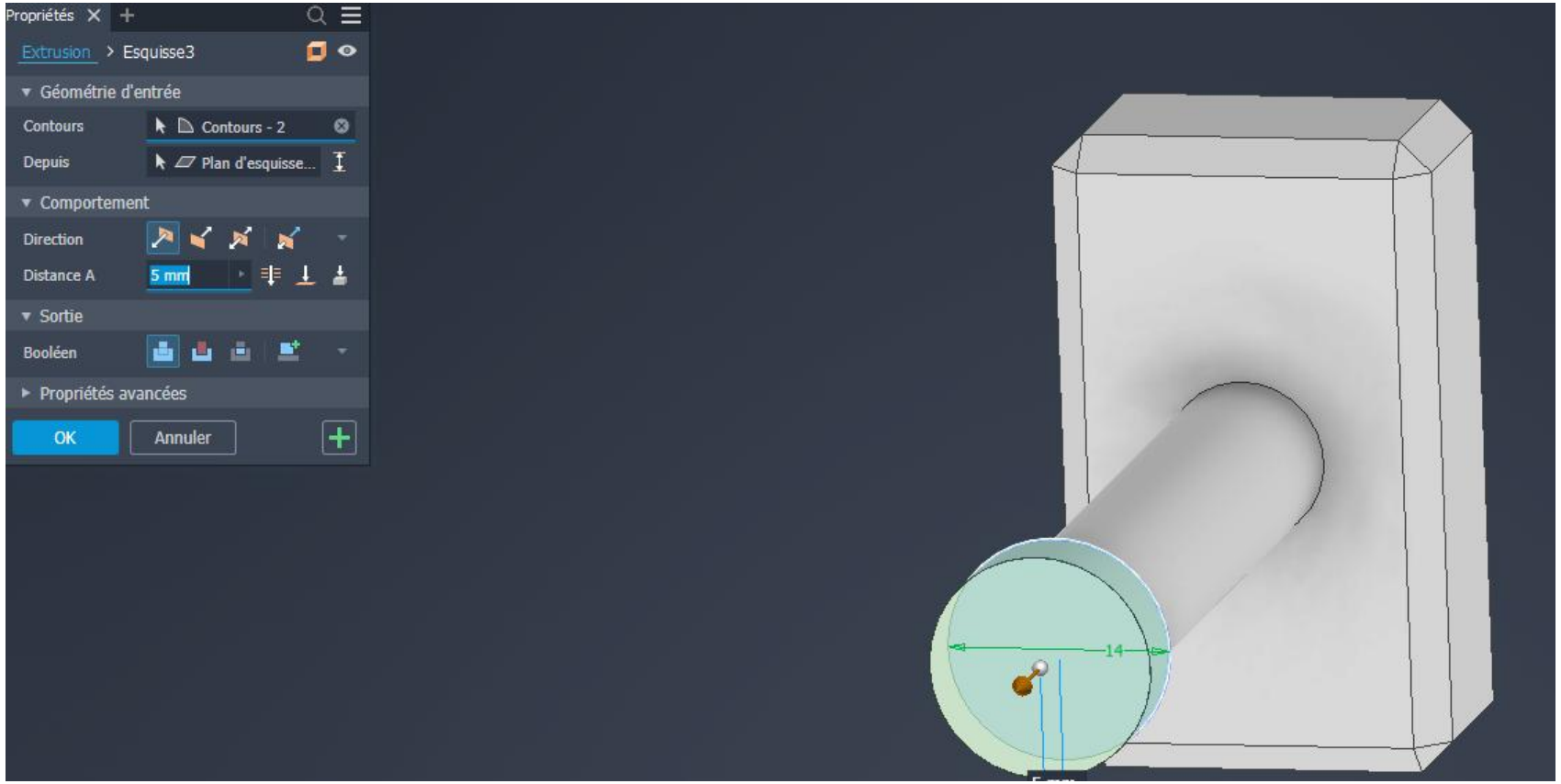
create a circle
with a diameter
of 14



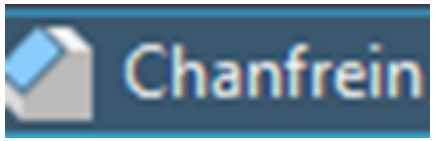
Step 12:



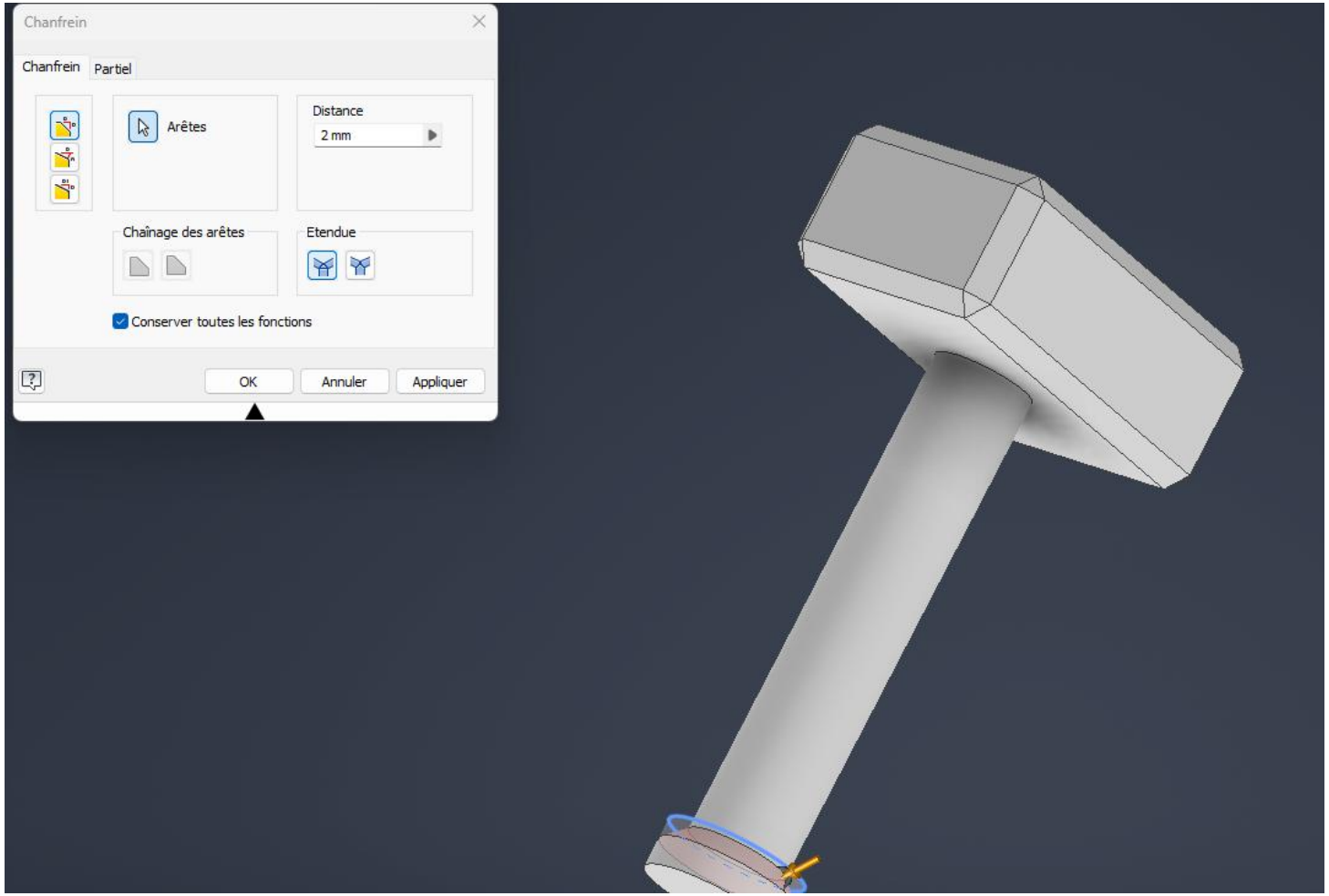
create an extrusion of 5



Step 13:



Select the edge and make a chanfrein of 2



Step 14:





Select the weight of the hammer and go to 'default' and select plomb



Step 15:





select the handle of the hammer and go to 'default' and select metal 1400F rouge



Step 16:





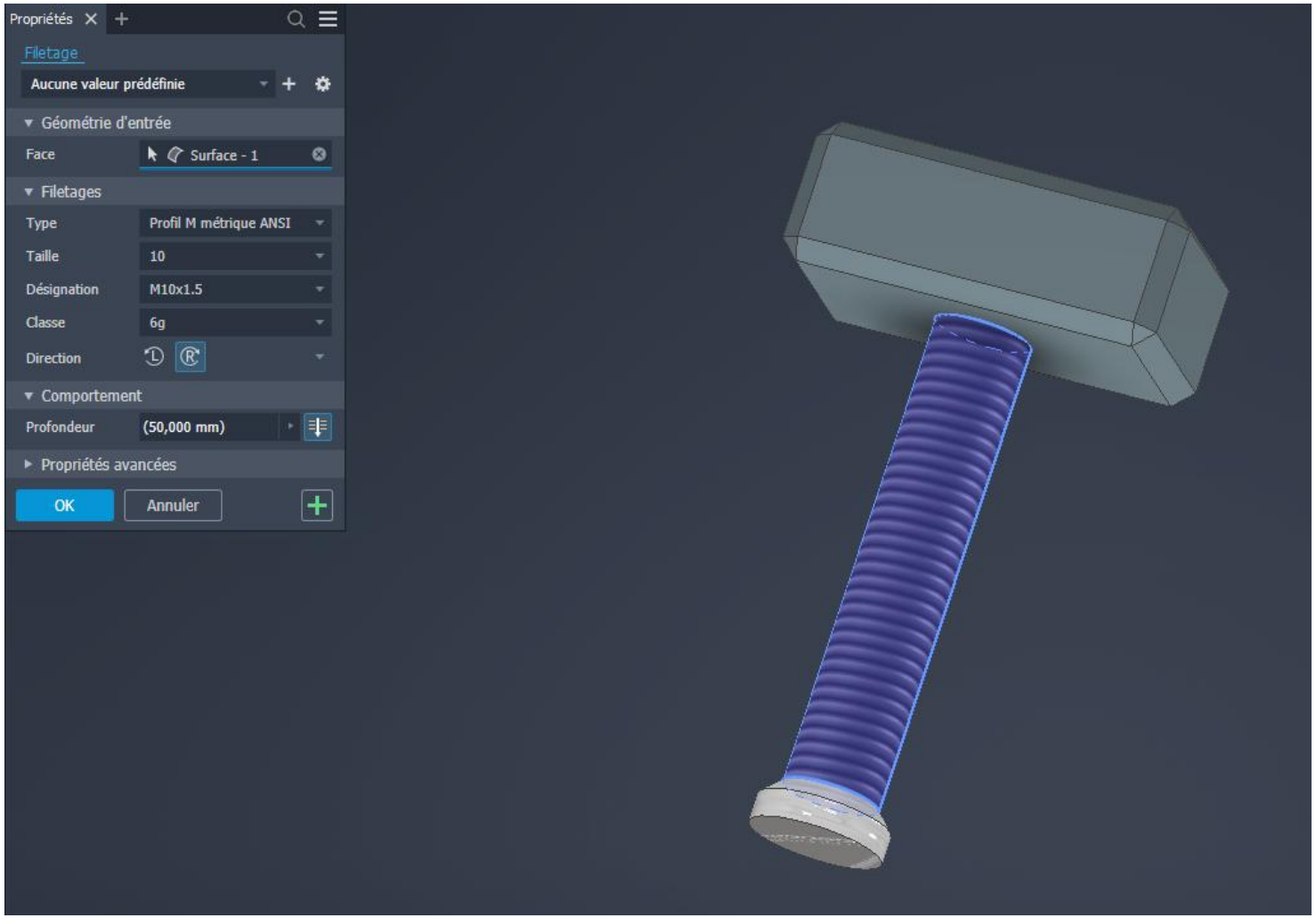
select the extremity of the hammer and go to 'default' and select poli plaque losange



Step 17:

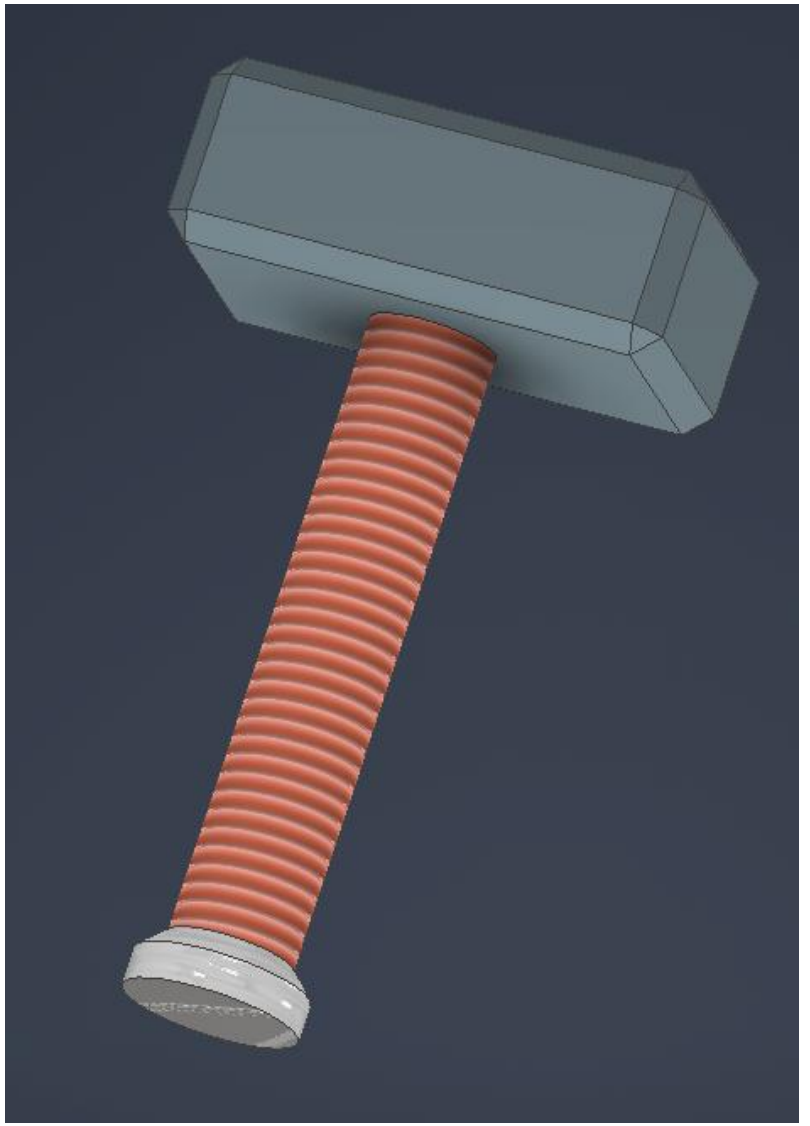


select the handle of the hammer and select filetage



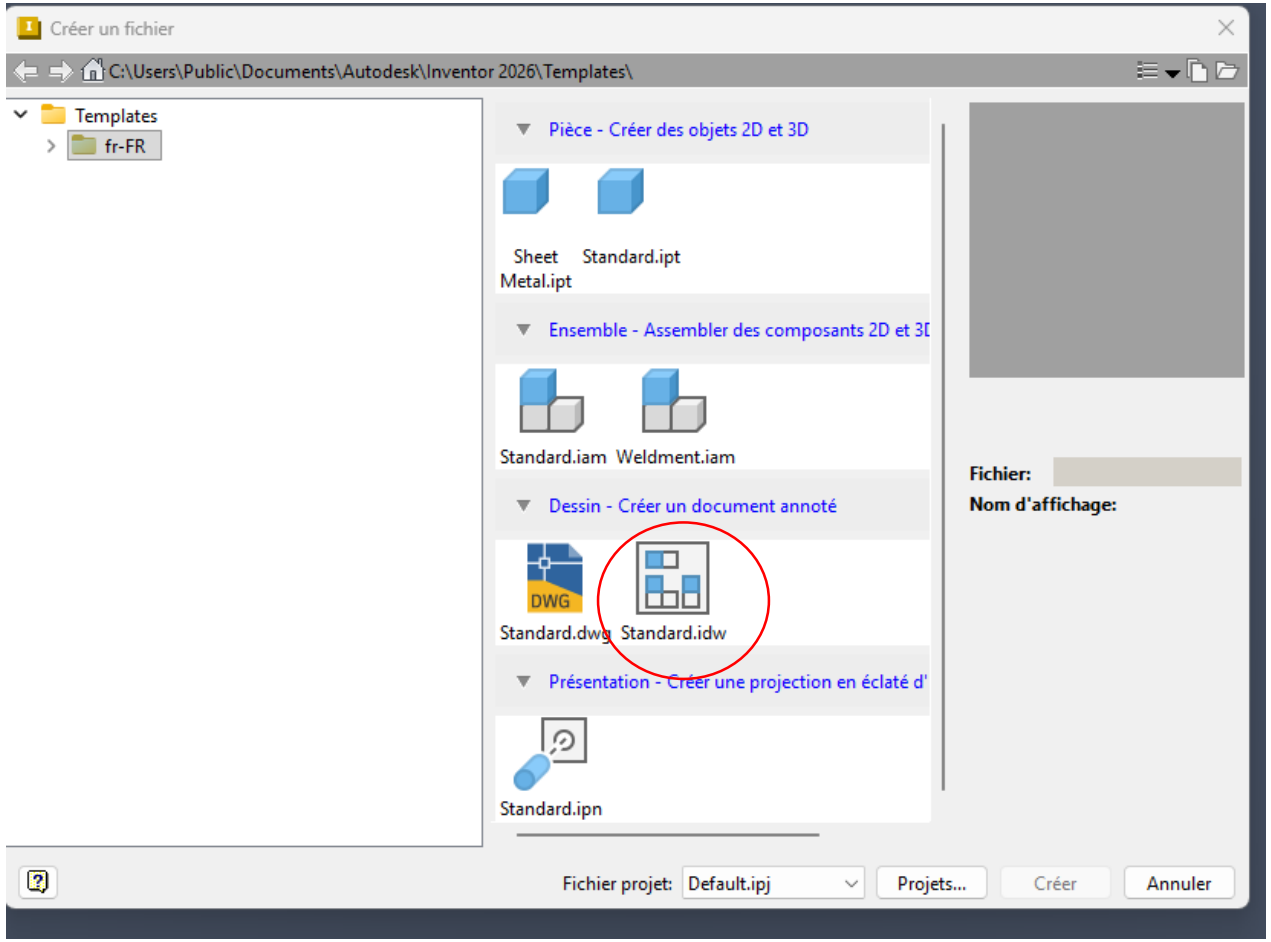
Step 18:

you finished your hammer!!!



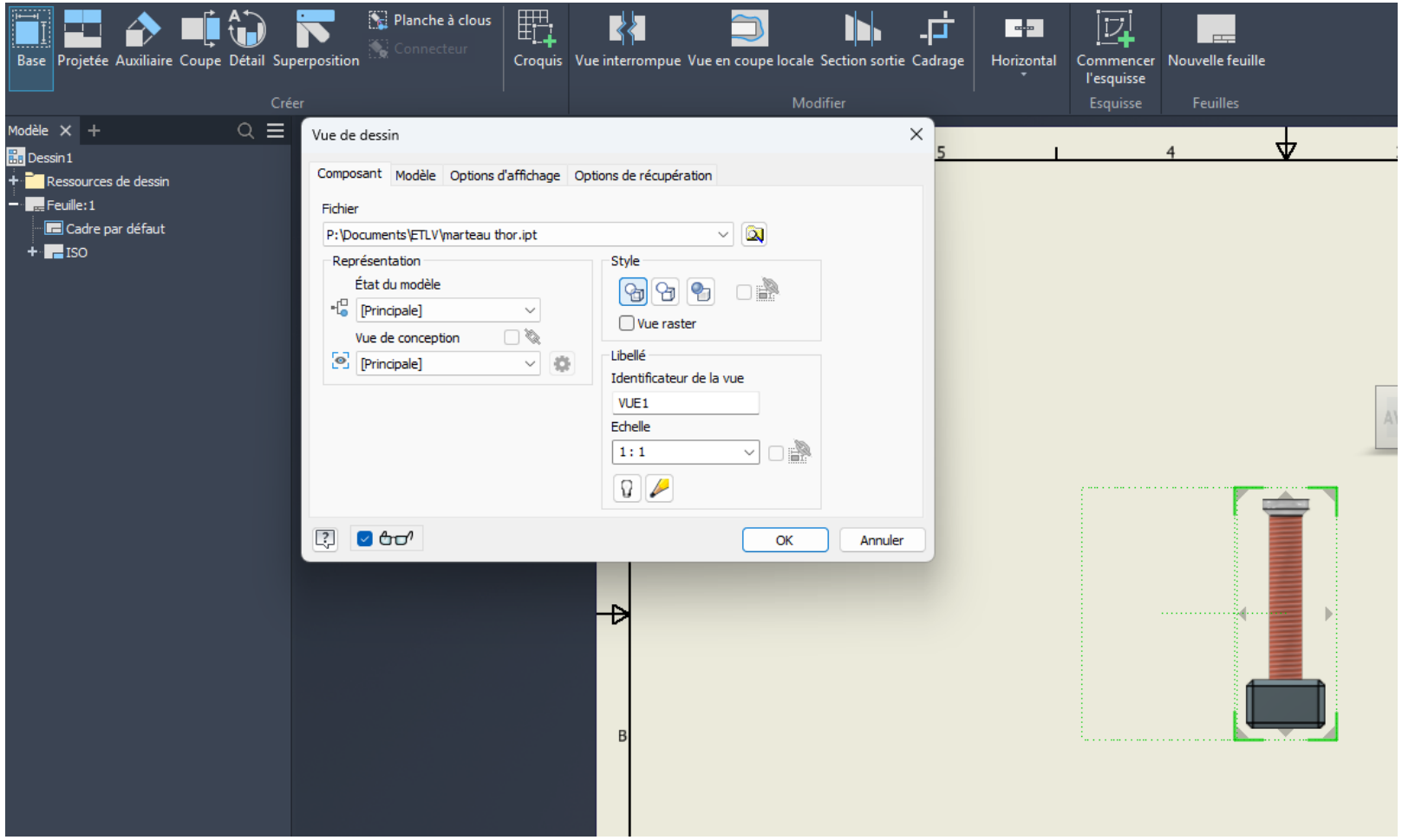
Step 19:

Save your hammer and create a new drawing file to make an assembly drawing and don't forget that you shouldn't close your part



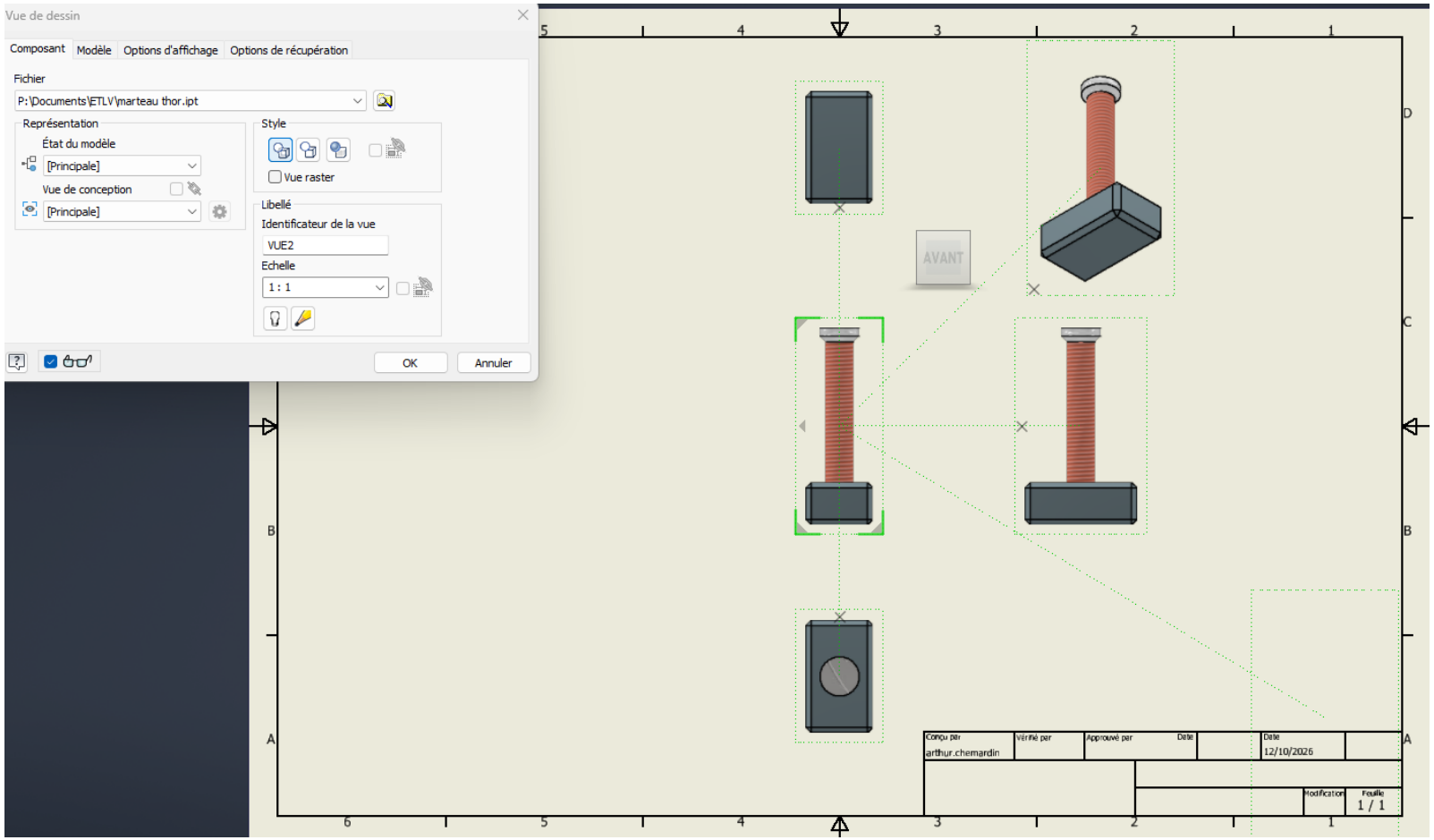
Step 20:

Go to the base and your piece will appear on the drawing

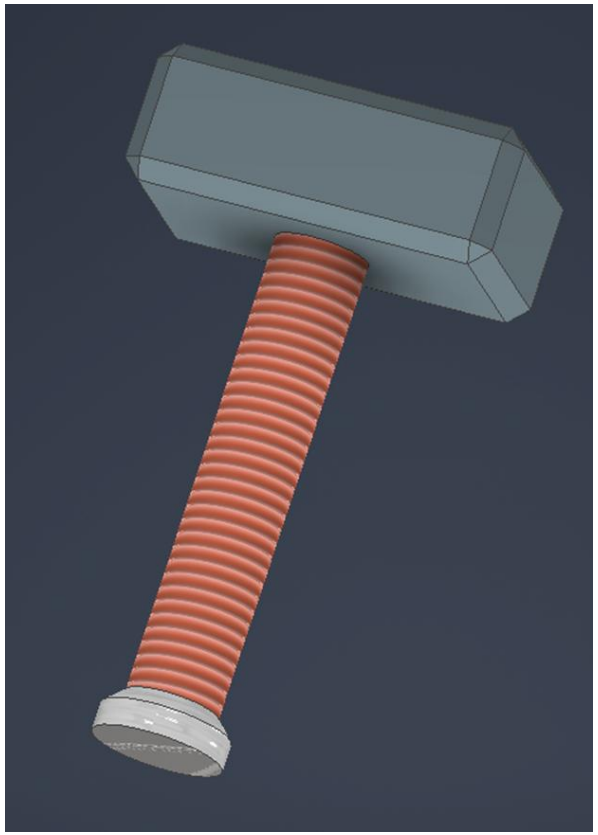


Step 21:

place the
different views



Step 22:



You are finished the hammer and theits overall drawing

