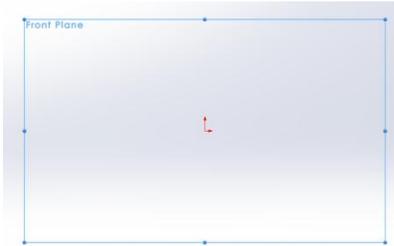
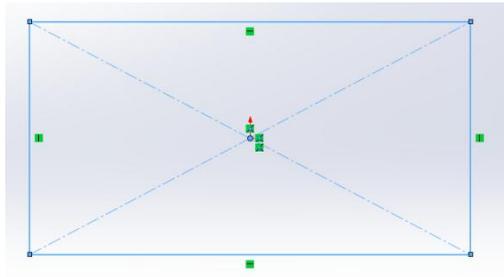


This is how I would build the TOE CLAMP from Homework 1. Please note I am making up dimensions as I go along. Also note that I am incorporating “added techniques” in an attempt to show you what is possible, to encourage you to experiment and develop your skills and knowledge. And my own *personal preference* is to *not* work normal to the drawing face.

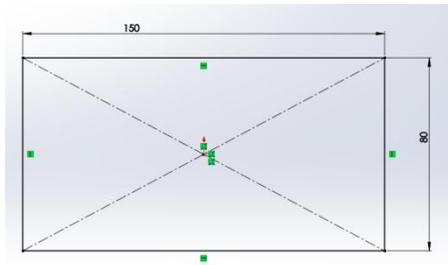
First, pick the FRONT plane to sketch.



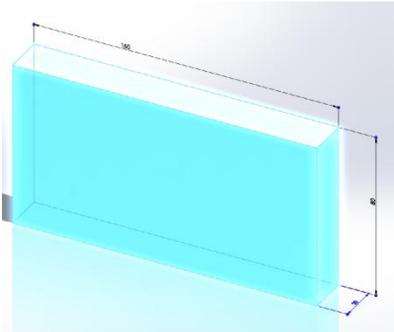
Draw “centerpoint rectangle” centered on the origin.



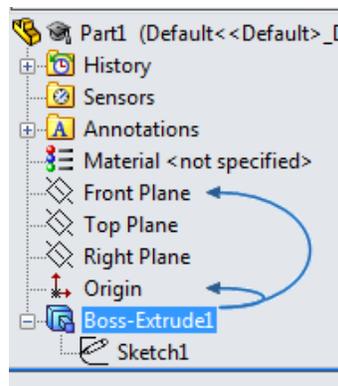
And dimension it. Note how everything is black, indicating that SW says it understands the shape fully. I.e., “fully defined”.



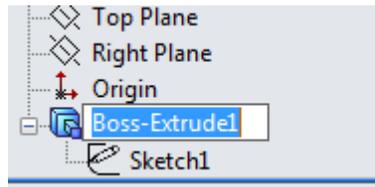
And extruded 20 mm to form my first Extruded Boss/Base.



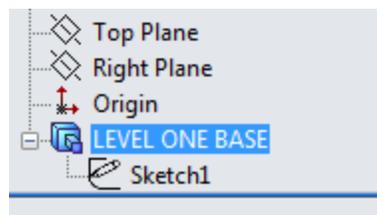
Now before I go any further, take a look here:



If I go to the Boss-Extrude1 and slowly click (not rapid doubleclick) I get this:



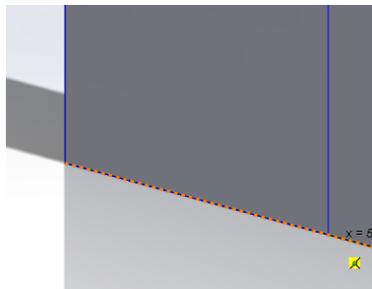
I now rename that feature:



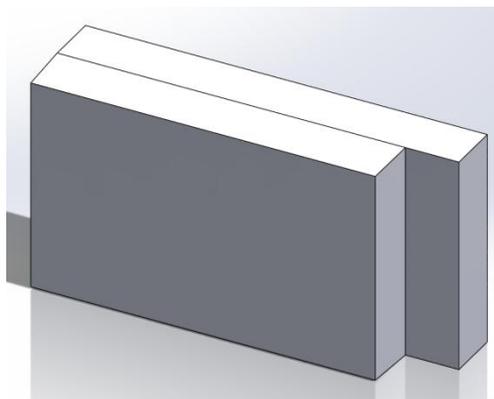
Renaming your features can seem like a hassle. Trust me, in a large model, it's a life-saver. On to LEVEL TWO. Pick the first point; watch for the orange "I got the vertex" circle.



And pick the line for the other rectangle vertex; watch for the "I got the line" orange highlighting:

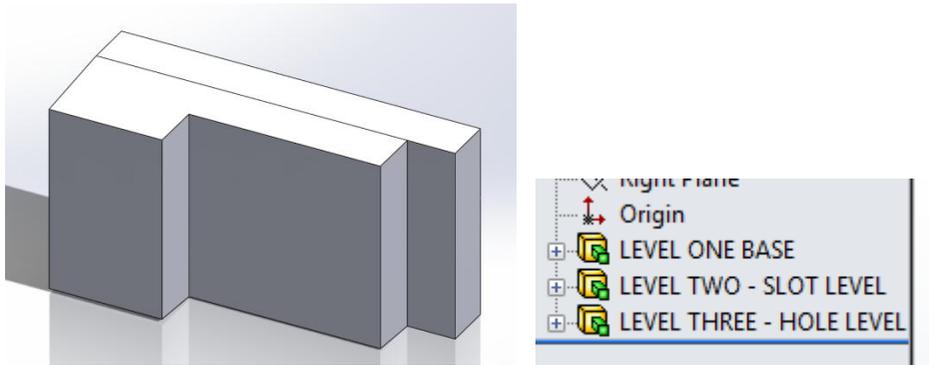


And add my dimensions, and again extrude 20 mm. Not forgetting the 2 degree draft angle.

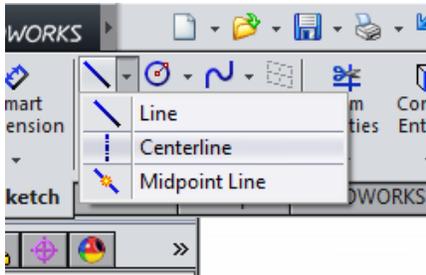


Rename the feature LEVEL TWO – SLOT LEVEL.

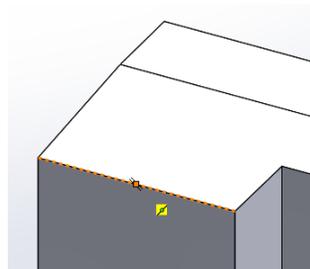
And then do LEVEL THREE – HOLE LEVEL.



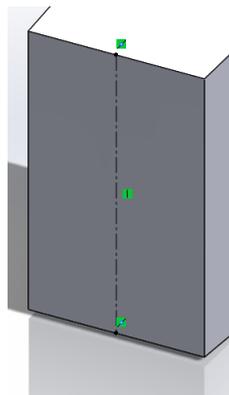
Open a new sketch on the face for the hole. Go to LINE, then CENTERLINE:



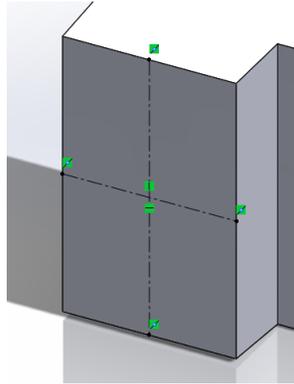
Hover over the top line near the center until it turns orange and you see the midpoint appear:



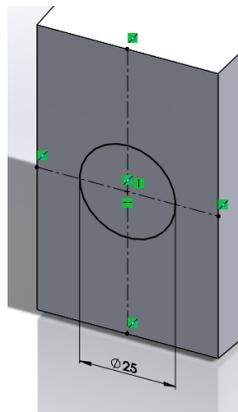
Draw a line straight down until the next edge. Again, hover over it near the middle until the midpoint appears, then click on it to draw the line, then hit ESCAPE to end the LINE command.



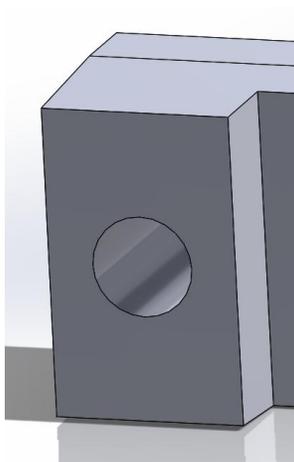
Repeat for the horizontal of the crosshairs.



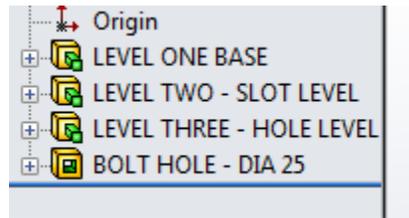
Now, I can put the circle on the crosshair center. Why? My **DESIGN INTENT** is to make sure that hole is centered on that face. No matter what changes get made to the dimensions, that hole will stay centered. Also, by doing it this way, anyone who comes along and looks at the sketch should understand what I intended.



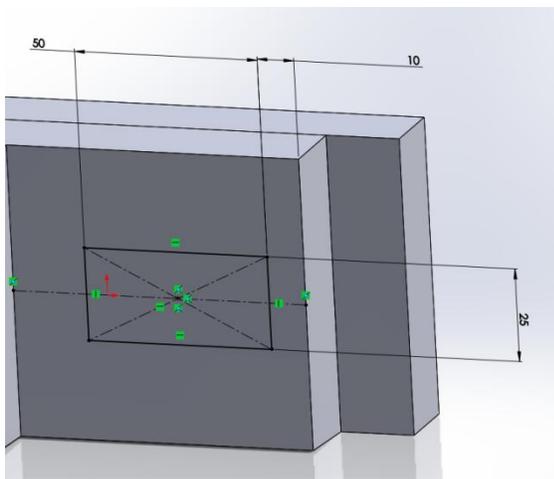
And make my hole. (Adjusted my positioning of the part on screen.)



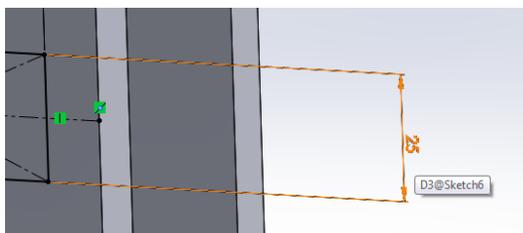
And rename the hole. Note that I add both function and dimension.



Now, I only add *one* centerline for the slot. Why? Because in this case I want the slot centered up-and-down, but... *as an example*... I do *not* want it truly centered. Let's say, for the sake of argument, that I always want one end of the slot 10 mm from the edge, and 50 mm long. This way, no matter how the toe clamp gets lengthened, my **DESIGN INTENT** is maintained.

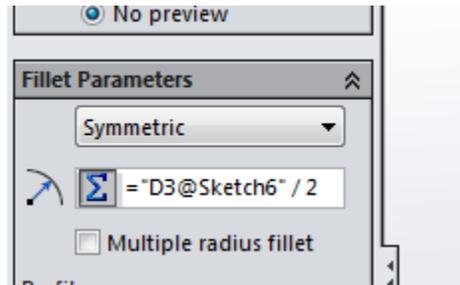


Now, re-open the sketch for this slot rectangle, and hover over the 25 mm slot width.

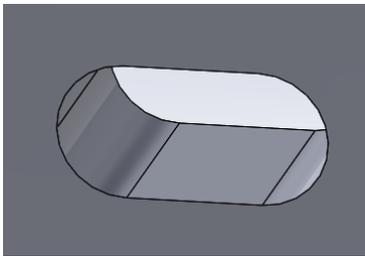


Each dimension has a name. Take note of this, **D3@sketch6**, then exit the sketch. Don't forget to rename your slot. In my case, I just said SLOT RECTANGLE.

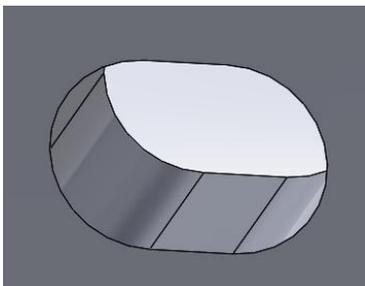
Start your FILLET tool, pick the four slot corners. Now look at what I'm entering for the fillet radius:



The format for this is `= "d3@sketch6"`. Yes, the quotation marks are included. Why would I do this? **DESIGN INTENT.** I want those radii to be full rounds on the end of the slot no matter how the slot gets changed. First, make my fillets. And name the fillet **SLOT FULL ROUND FILLET**.



What do you think would happen, now, if I changed the slot width to 35 mm? I'll go into my sketch and make the change.



See how the fillets track with the slot width? Now, I'll change it back. Do the corner fillets around the outside. Final part. I'm not going to do the annotations though.

