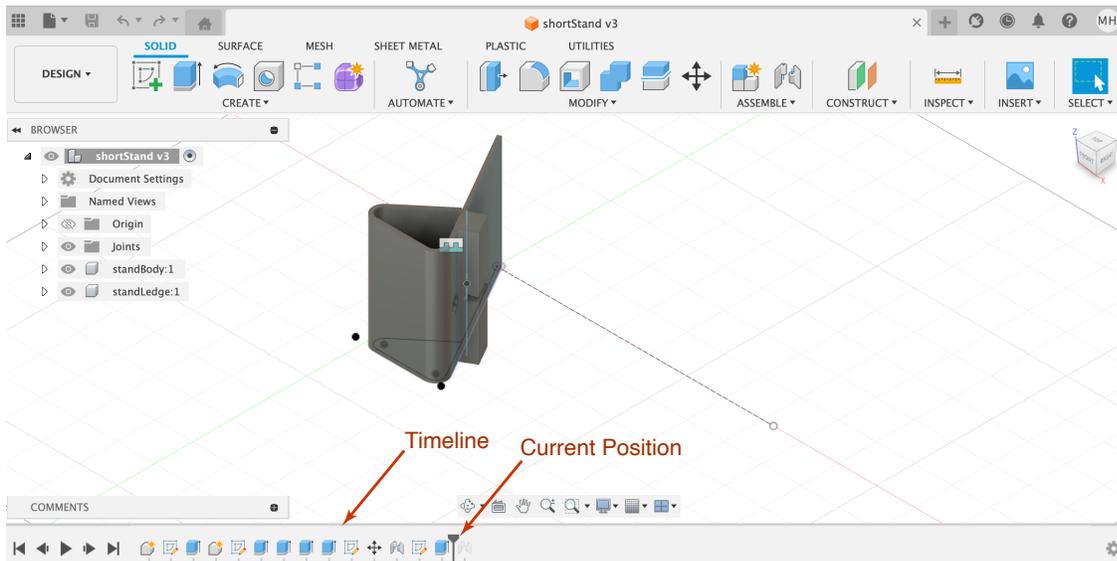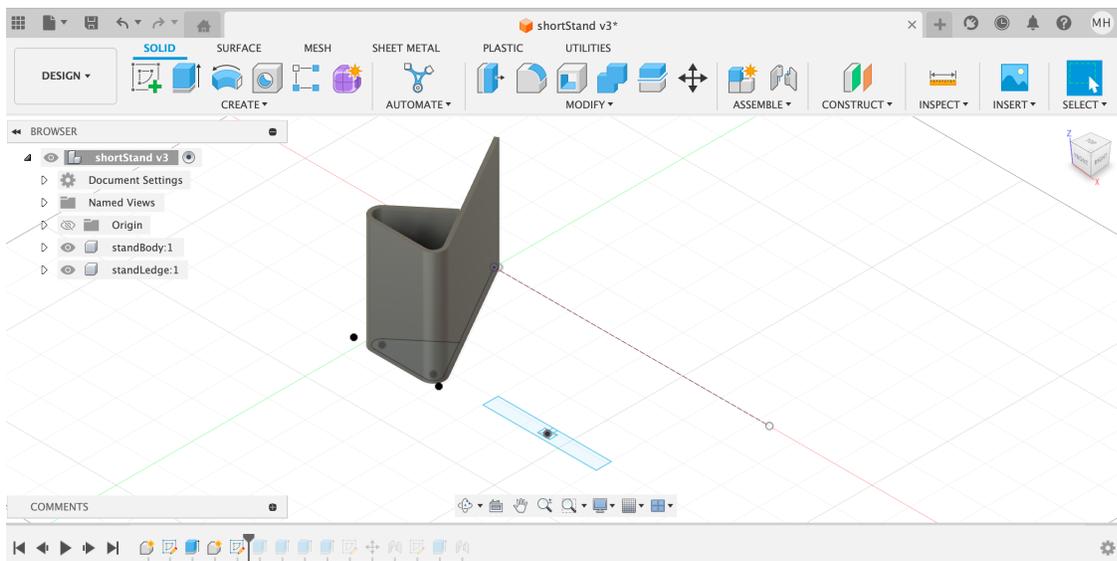# Modifying a Fusion 360 Design

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There are a number of different ways to modify a design in Fusion 360. One way is to open up the original sketch that generated the body in the Browser (LHS of window) by double-clicking on its name. Another way is to modify the body itself (*e.g.,* using the body filleting tool). One of the easiest ways is often the most overlooked way: the timeline. At the bottom of a Fusion 360 design (see below) you will see a number of icons with a little "current position" indicator at the end. Each of the icons represents one of your previous actions (*e.g.,* sketch creation, body extrusion).

For example, suppose I want to modify my phone stand by adding a fillet to the phoneLedge. I can scroll back in time to the point where I defined my sketch of that part as follows:

Once there, I can add some fillets to the front of the ledge. In doing so, Fusion 360 will generate an error warning, telling me that adding the fillet removed a constraint. This is because dimension constraints can only be applied to straight sections. The length of the now-rounded distance between the front and the back

of the phone distance cannot be dimensioned. Removing the original constraint is not a problem unless / until you want to change the ledgeWidth.

As mentioned in lecture, our design is technically underconstrained. This is not a problem; however, underconstrained items can change when you change something (*e.g.,* your parameters). This can create problems under certain conditions. Having said that, there are very few Fusion 360 designers who produce fully constrained drawings. Many designers don't even use formal dimensions; they just sketch what they want to 3D print.

Once you are finished modifying your original sketch, you simply slide the time back to end. This propagates your changes forward to the current time (*i.e.,* finished model).

In the case of our current design, there is no reason to prefer one modification method vs. another.