5 - Fastener Holes

5.1 - Clearance Holes

When you are designing with fasteners, you always have to pick an appropriate hole size so that the parts can assemble easily. A hole that allows a fastener to fit through easily is called a clearance hole. For rivets, this is very simple: the clearance hole diameter is equal to the diameter of the rivet, so a hole for a 3/16" rivet should be drilled with a 3/16" drill bit. For bolts, however, this is slightly more complicated.

The threads on a bolt actually extend slightly beyond the bolt's nominal diameter, so a bolt cannot fit through a hole that is equal to its nominal diameter. Therefore, a clearance hole for a bolt must be slightly larger than the nominal diameter, and industrial standards exist to define exactly how large that hole should be. For a 10-32 bolt, for instance, the standard clearance hole should have a 0.201" diameter, slightly larger than the bolt's 0.19" nominal diameter.

However, in Team 1732's shop, we do not have all the possible drill bit sizes, so we use the nearest fractional size instead. For a 10-32 bolt, we use a 13/64" bolt. This is equal to 0.203", so it is very close to the specified 0.201" size. In fact, for most bolts, adding 1/64" to the nominal diameter is accurate enough for a good clearance hole size.

5.2 - Taps and Tap Holes

A tap is a tool that cuts threads into a hole, effectively making it into a nut. This can be very useful where a nut cannot fit, or to reduce part count. Tapped holes also require a specific size of hole so that the tap has enough material to grab onto and create good threads. These holes, called tap holes, are also standardized, and for our purposes can be approximated to the nearest fractional size.

Figure 1: Tap and handle. The handle is turned to cut threads into the hole, much like driving a bolt into hole.
(image from Wikipedia.org)

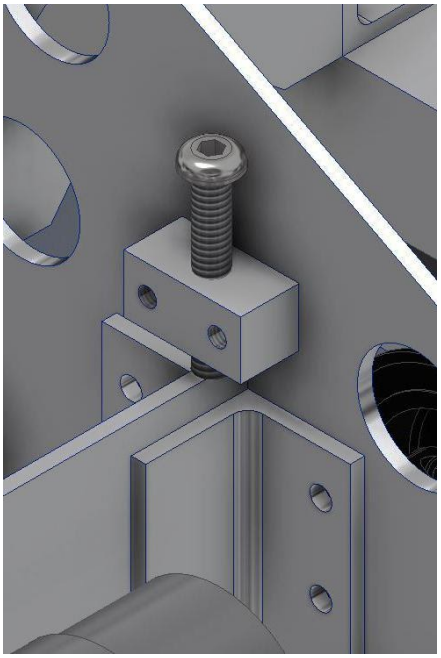

Figure 2: An example of a part with tapped holes from the 2018 robot. As the bolt turns, it extends and pushes the plate below it downwards. This could not have been accomplished with just a nut.
(image from Wikipedia.org)

5.3 - Modelling Clearance and Tap Holes in Inventor

Rather than consulting a table every time you need to make a clearance or tap hole, you can use the Hole tool in Inventor to do it automatically. The Hole Tool creates holes centered on points in sketches. It allows you to create many types of holes, with various seats and termination options.

The first icon under "Hole" is a simple hole. For this type, you can specify any diameter, as well as the direction and depth much like for an extrusion.

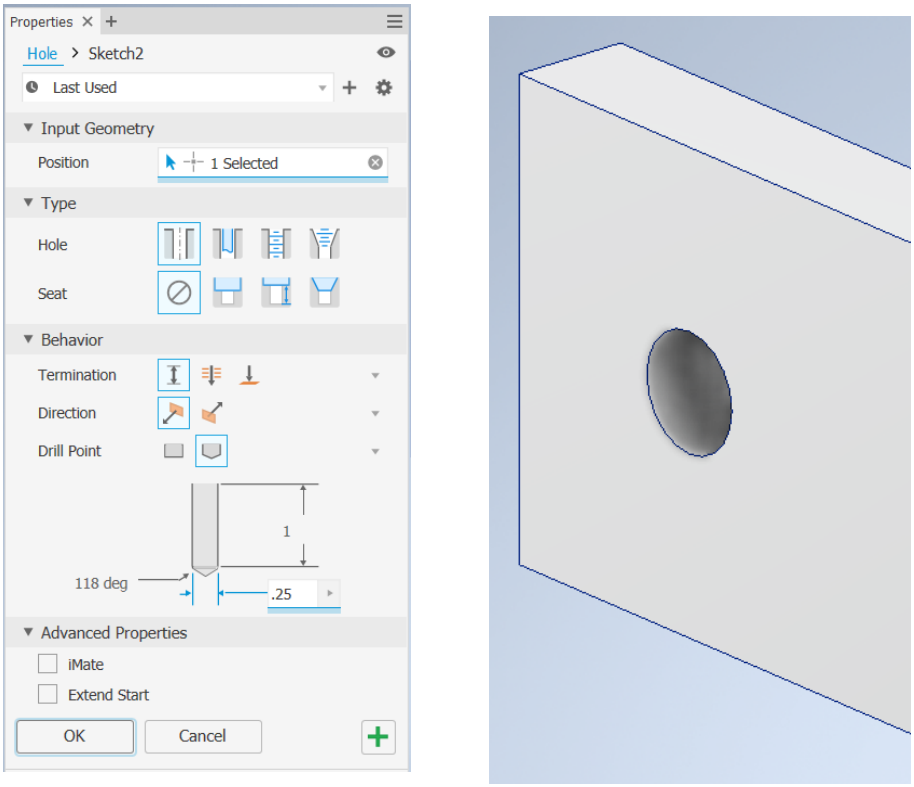

Figure 3: Hole tool for a simple hole

The second icon is for a clearance hole. For this type, you simply specify the screw size, and Inventor fills in the appropriate clearance hole diameter. There are options for the type of fastener and fit, but those are not necessary to get the correct hole diameter.

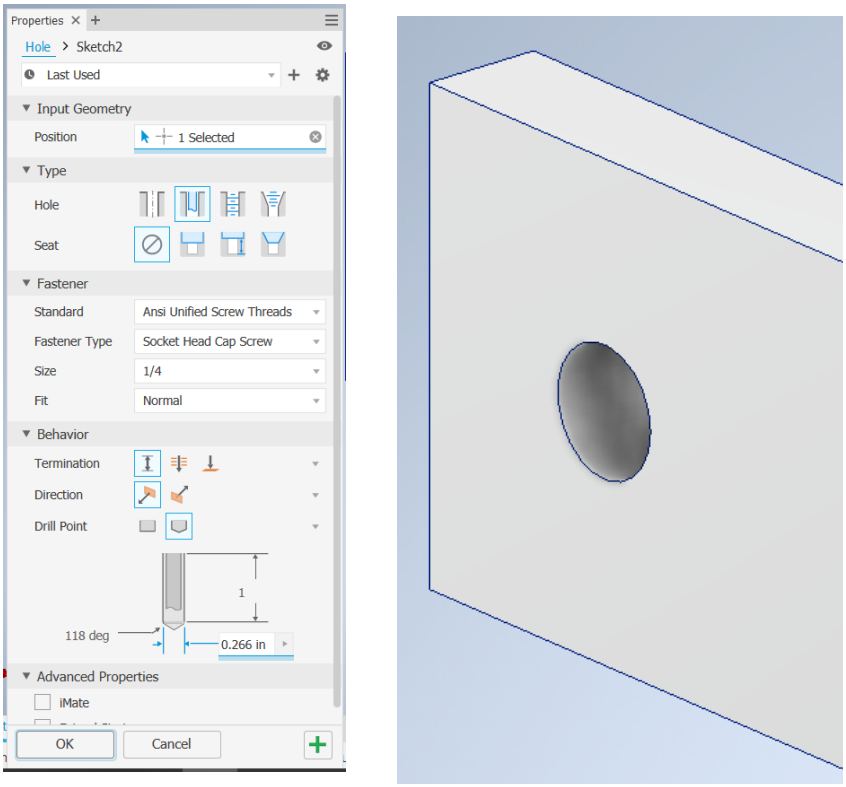

Figure 4: Hole tool for a clearance hole

The third icon is for a tapped hole. This works much like the clearance hole, where you need to specify a screw size, but you also must specify a designation, which indicates whether it is a coarse or fine thread. Then, there is also an option at the bottom to set the length of the threads, as the threads do not always extend the whole length of the hole. Rather than computing the complicated profile of threads, Inventor creates a cosmetic thread, changing the appearance of the hole surface to look like it is threaded.

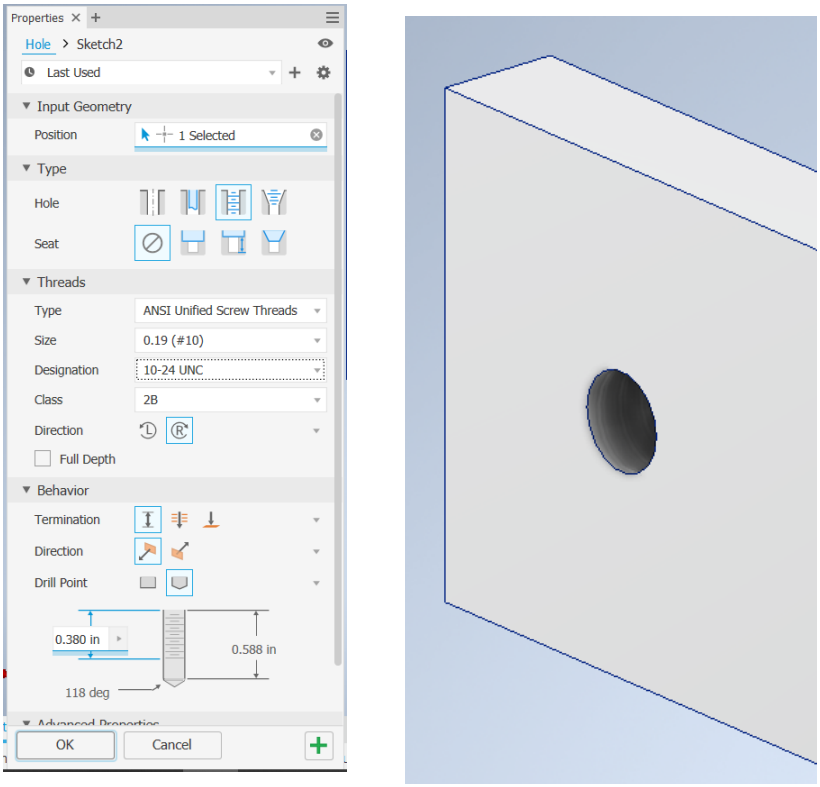

Figure 5: Hole tool for a tapped hole. Note the slightly darker hole surface due to the cosmetic thread.

The fourth hole icon is for a tapered tapped hole. These are used for things like pipe fittings, so they are not necessary for FRC design. The row of icons below hole type is for the seat type, which is where you can set a countersunk hole (the fourth icon) if you are using countersunk bolts. The first option, "None", is sufficient for most holes, however. The second and third, "Counterbore" and "Spotface", are not necessary for our designs.

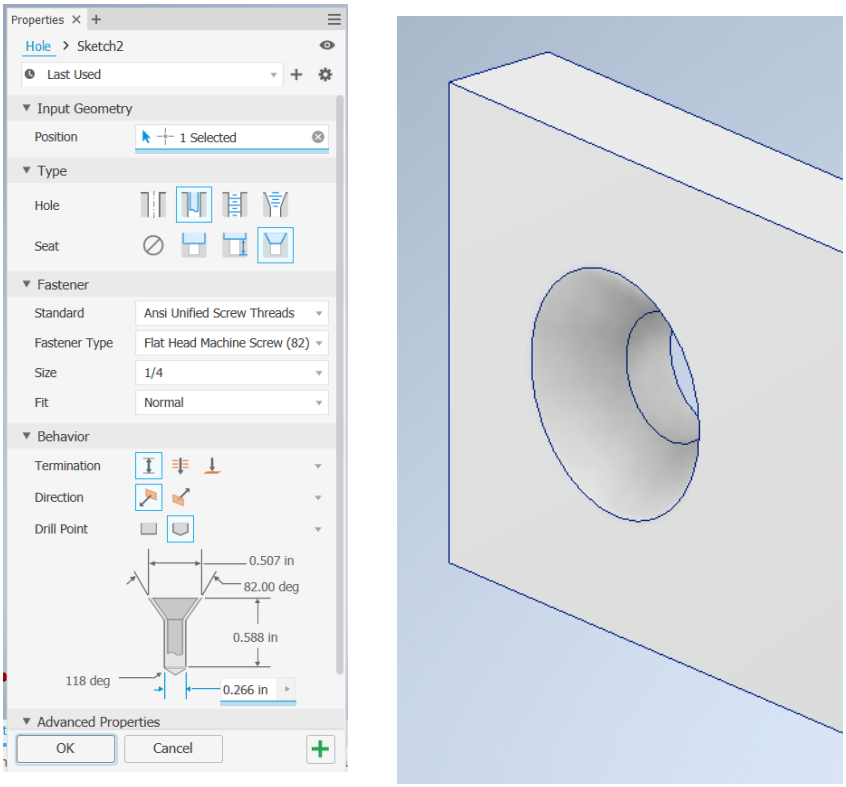

Figure 6: Hole tool for a countersunk clearance hole

Design Challenge 5: Hole Sizes

In Inventor, model a simple 1/4"-thick aluminum plate that is at least a 3" square. Then, anywhere on its surface, place holes for the following fasteners. Even though the location of the holes are arbitrary, remember to fully constrain your sketches.

1. 3 x 5/32" rivet, in a line pattern 0.5" apart
2. 1 x 3/16" rivet
3. 2 x 10-32 bolt, 1.5" apart (clearance)
4. 1 x 5/16-18 countersunk bolt (clearance)
5. 1 x ¼-20 bolt (tapped)