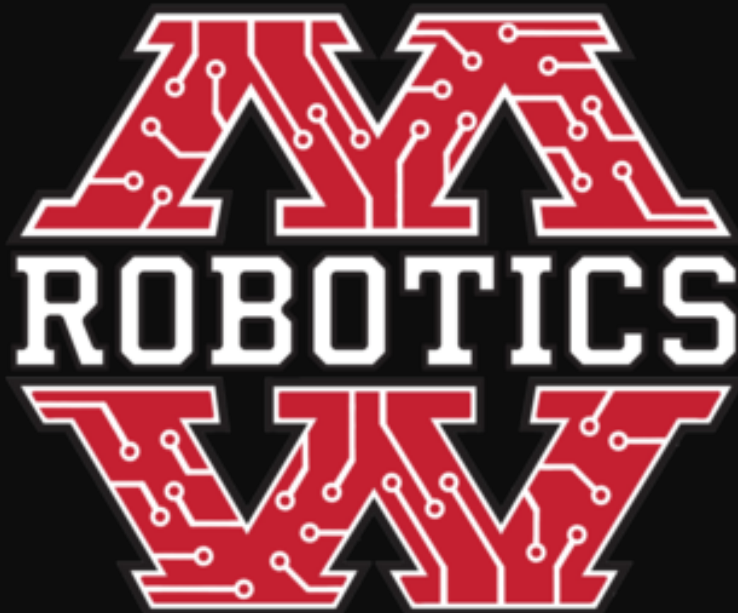


Improved VEX Universal Joint

By 8110R “The Knights”



PDF Report

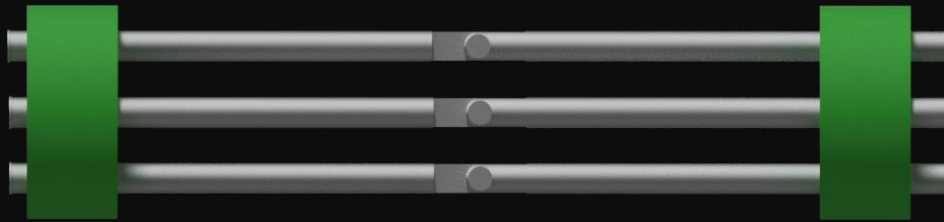
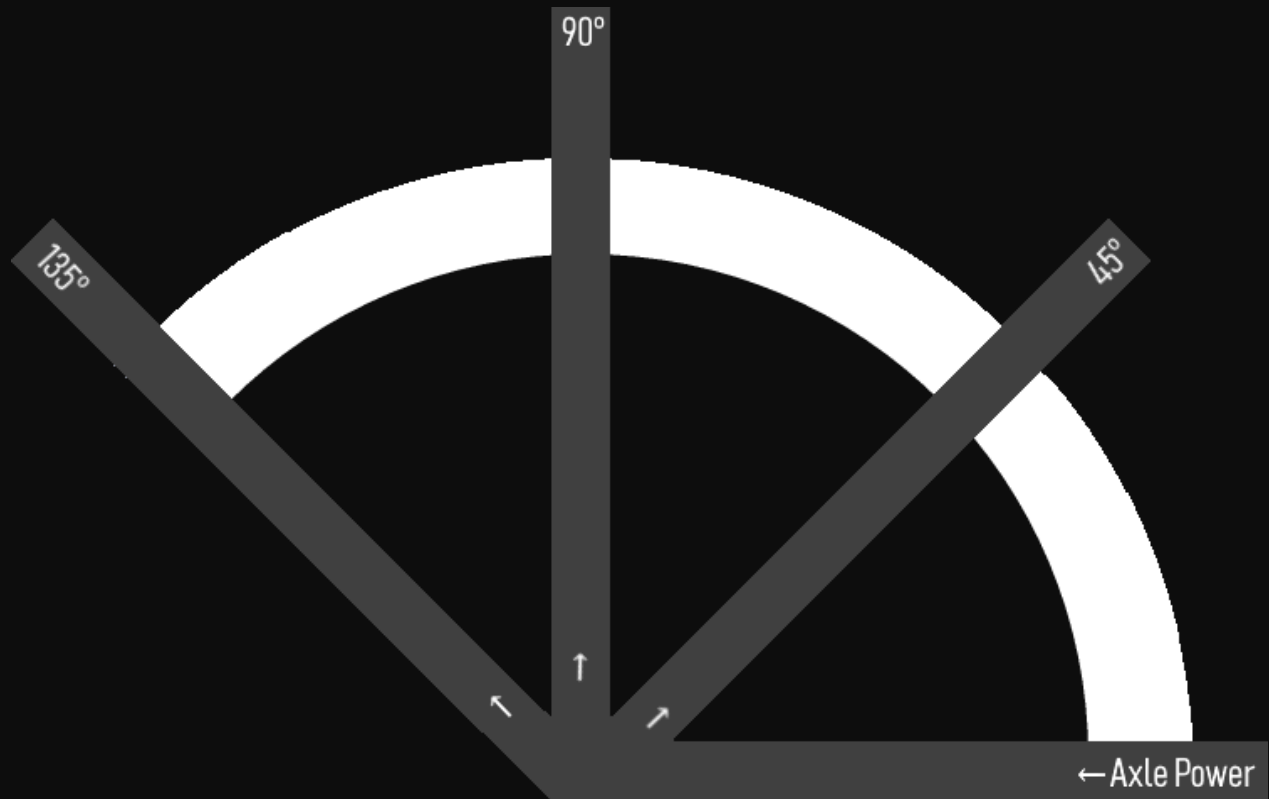


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Why it was made

This part was created with the goal of improving the transfer of axle power up to or beyond the current range of standard VEX universal joints.



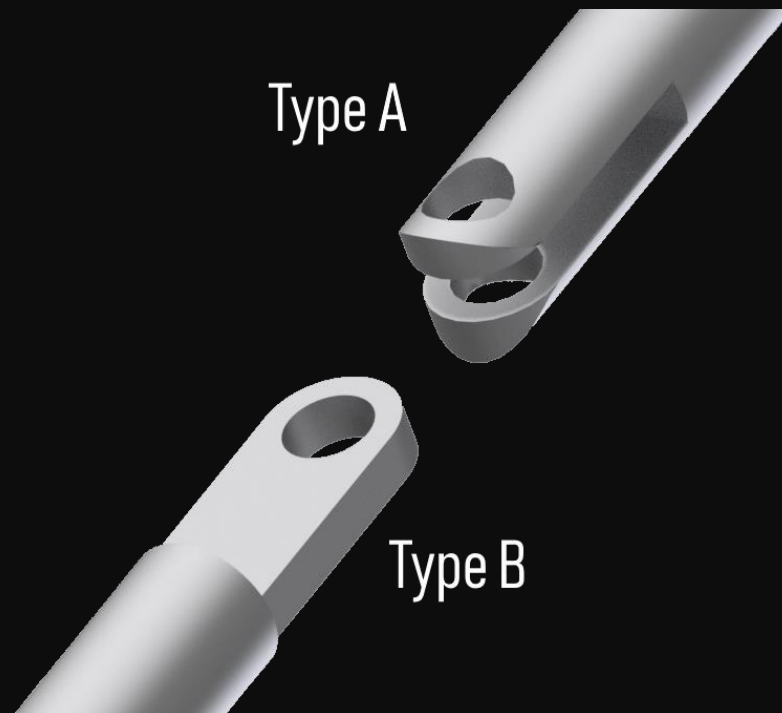
The regular VEX joints draw the most inspiration from standard universal joints. While this does allow for the transfer of axle power with non-parallel shafts, there is a small range of motion at which this can be accomplished. Anything close to or past 90° wouldn't work, which means you'd need multiple joints. This takes up valuable space, important with size limitations. This led to the creation of a new joint that transfers axle power past the 90° mark.

How it works

This joint connects to two axles with bases that go on the ends in a similar manner as the standard joint, but with a different connector.



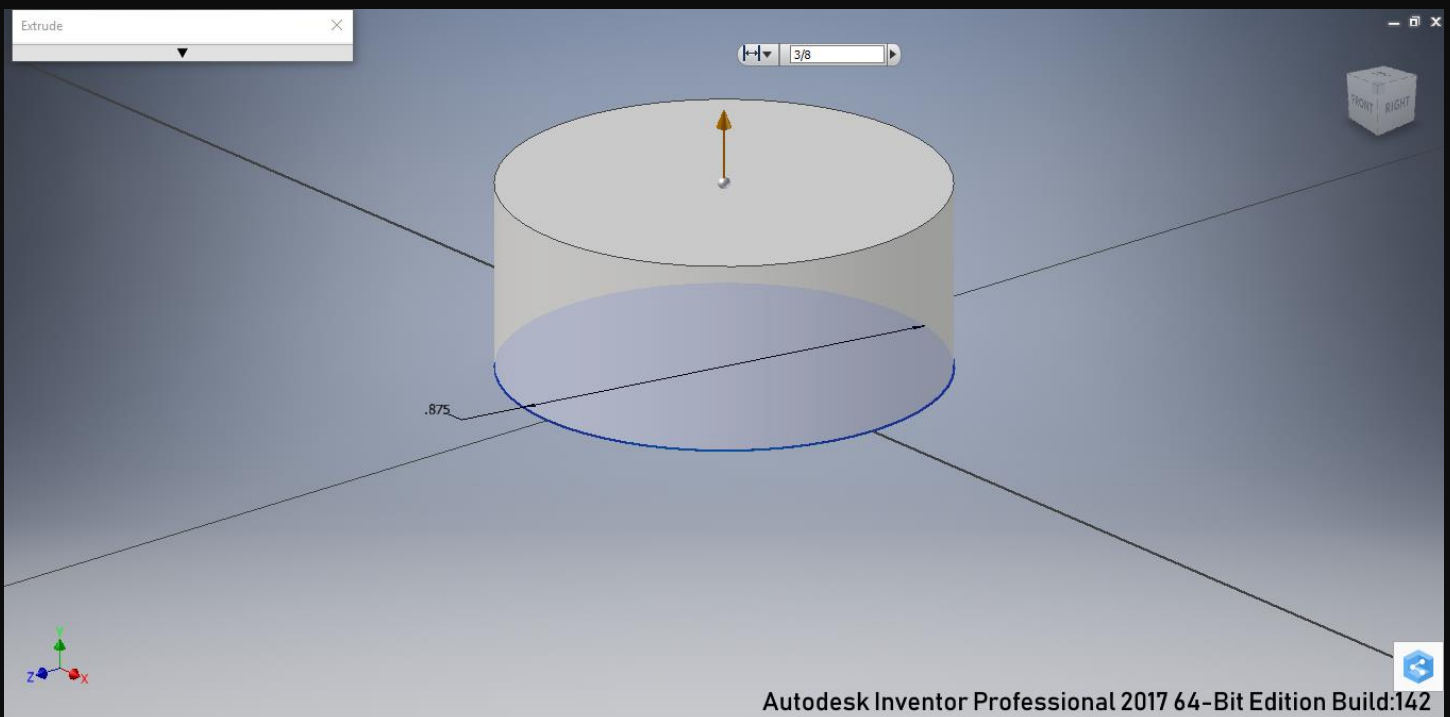
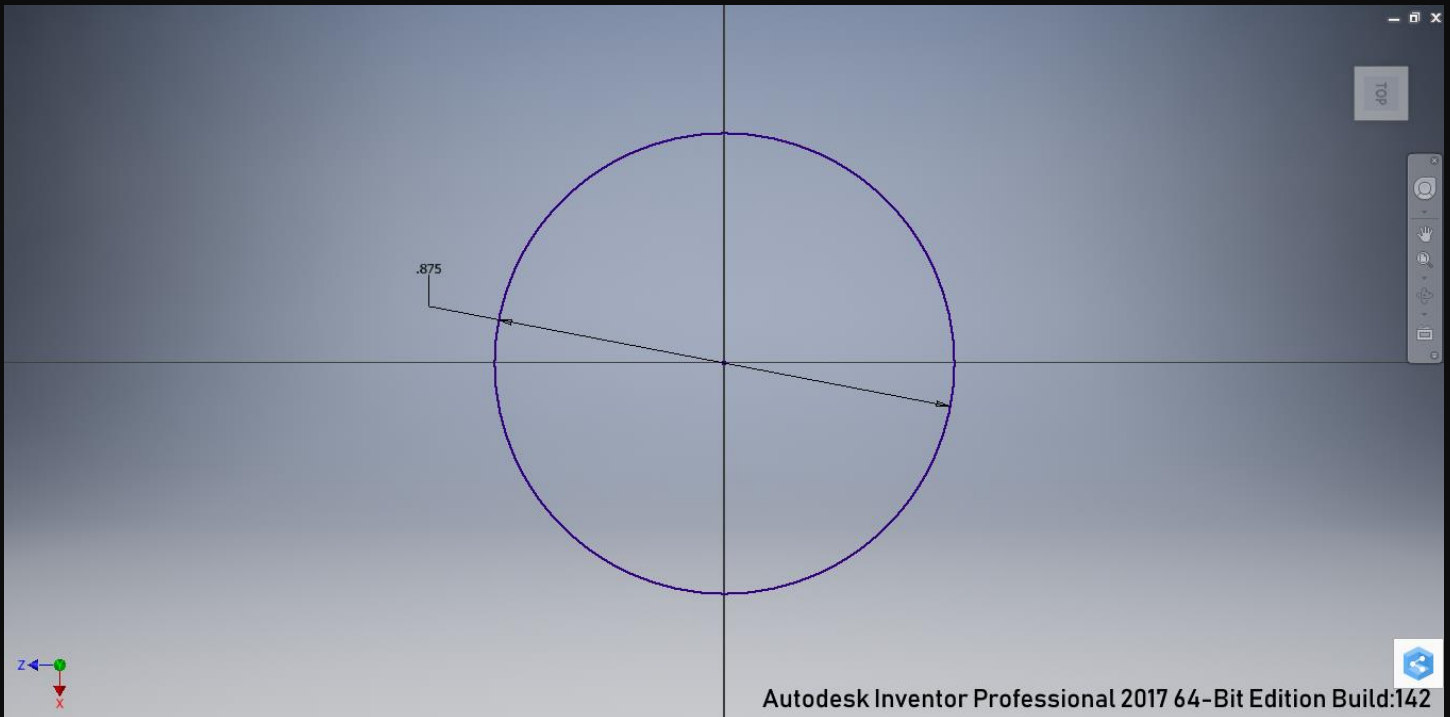
This connector is composed of three rods in each of the three holes of a base. The rods are either type "A" or "B", and will match up with the other base's rods.



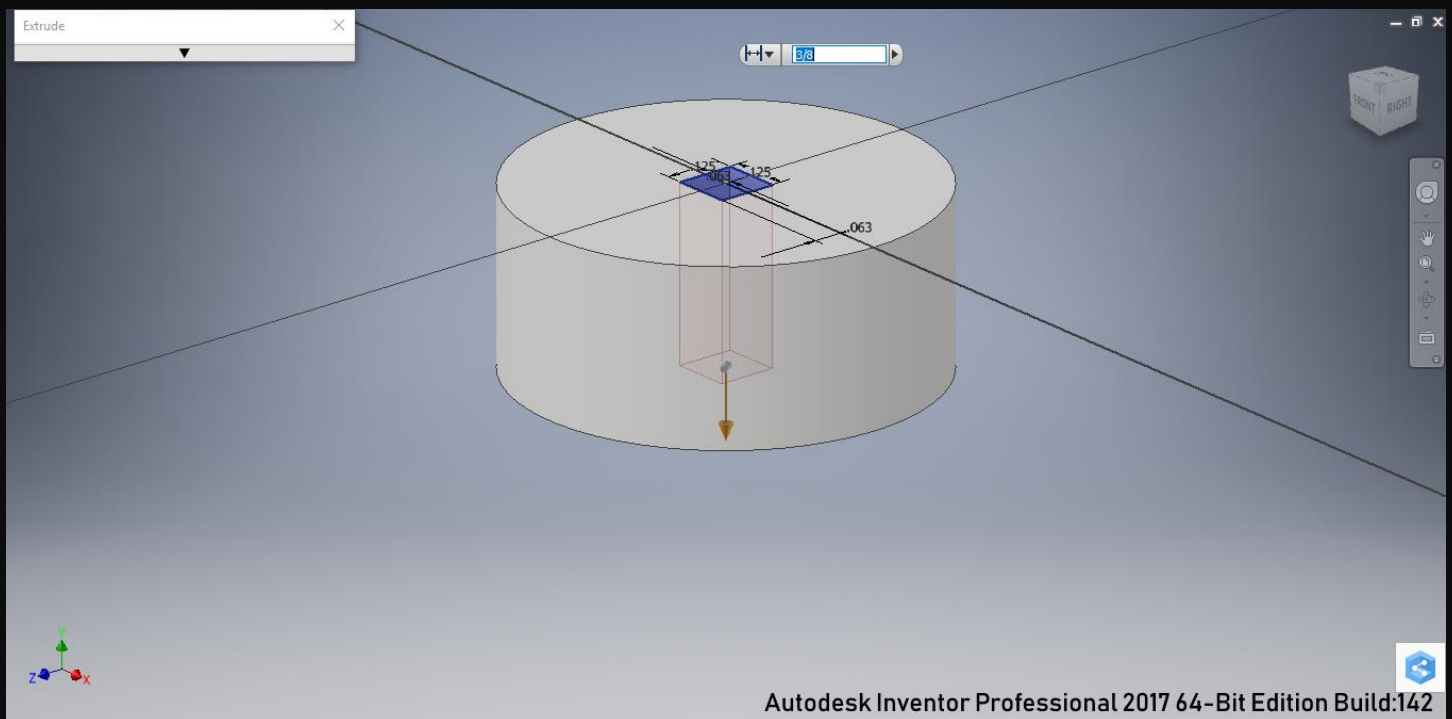
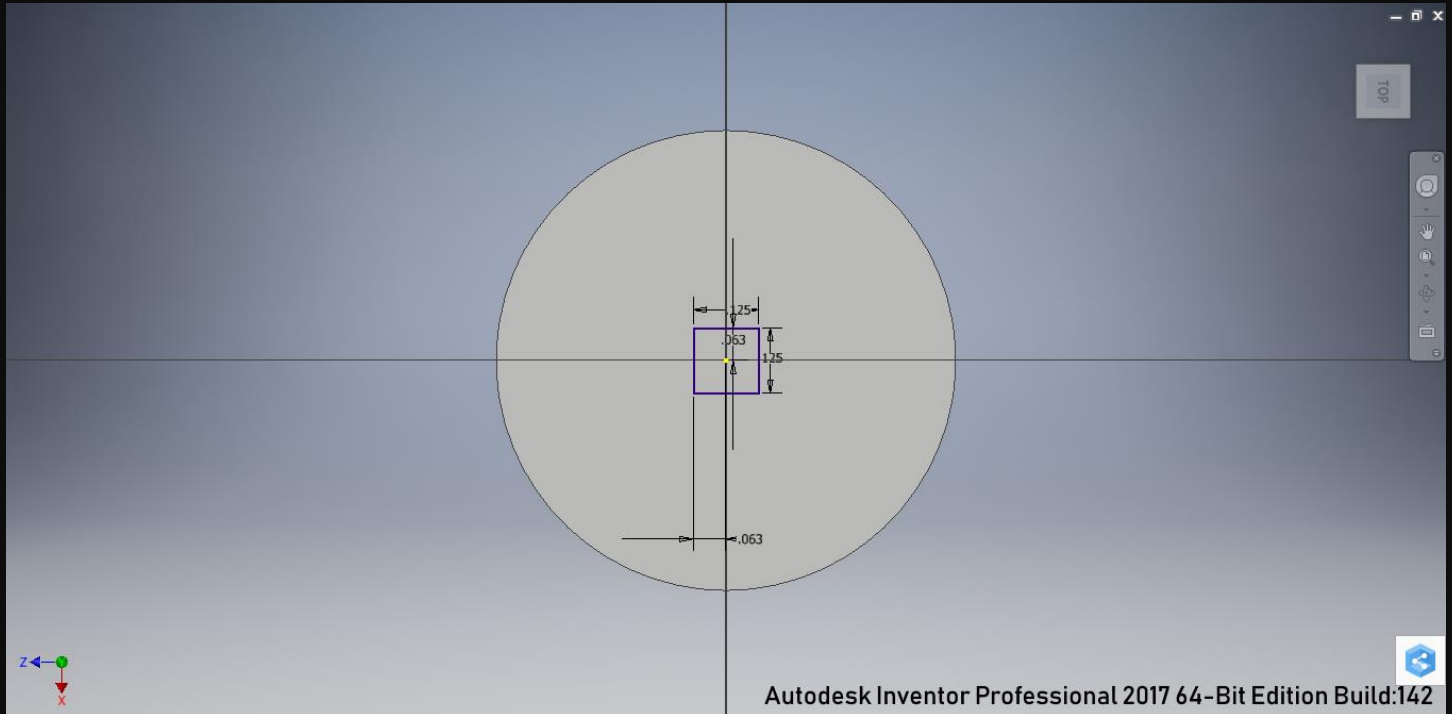
The rods are allowed to slide through the holes freely. This allows for the bases to turn together by having the rods shift through the holes, while being able to keep their angle. The three connections each being composed of two rods that can turn allow for the angle of the joint to be adjusted. The design of the rods were designed with extra gap space in the middle to allow for the rods to be adjusted for angles that extend past 90°.

How it was made

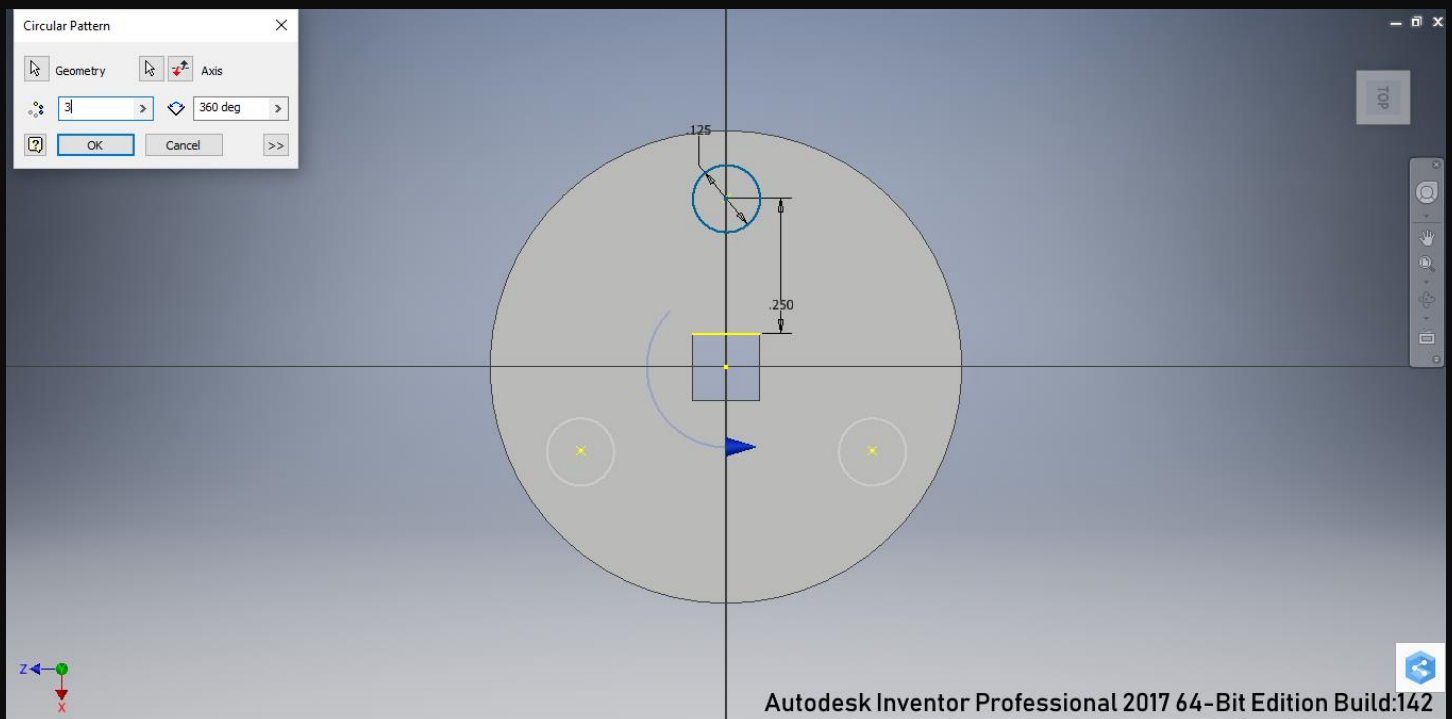
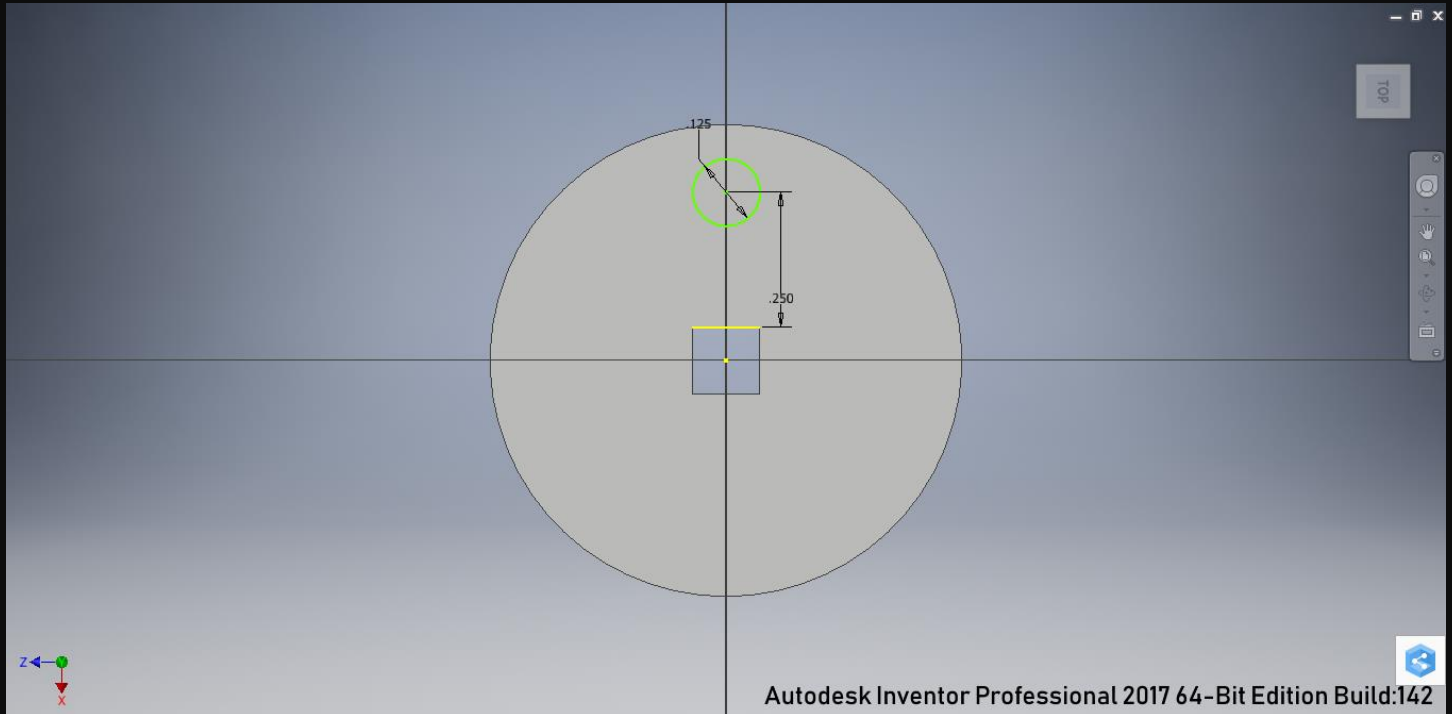
This component was made by first creating the base parts. A cylinder was first sketched out in inventor, then extruded to the desired length.



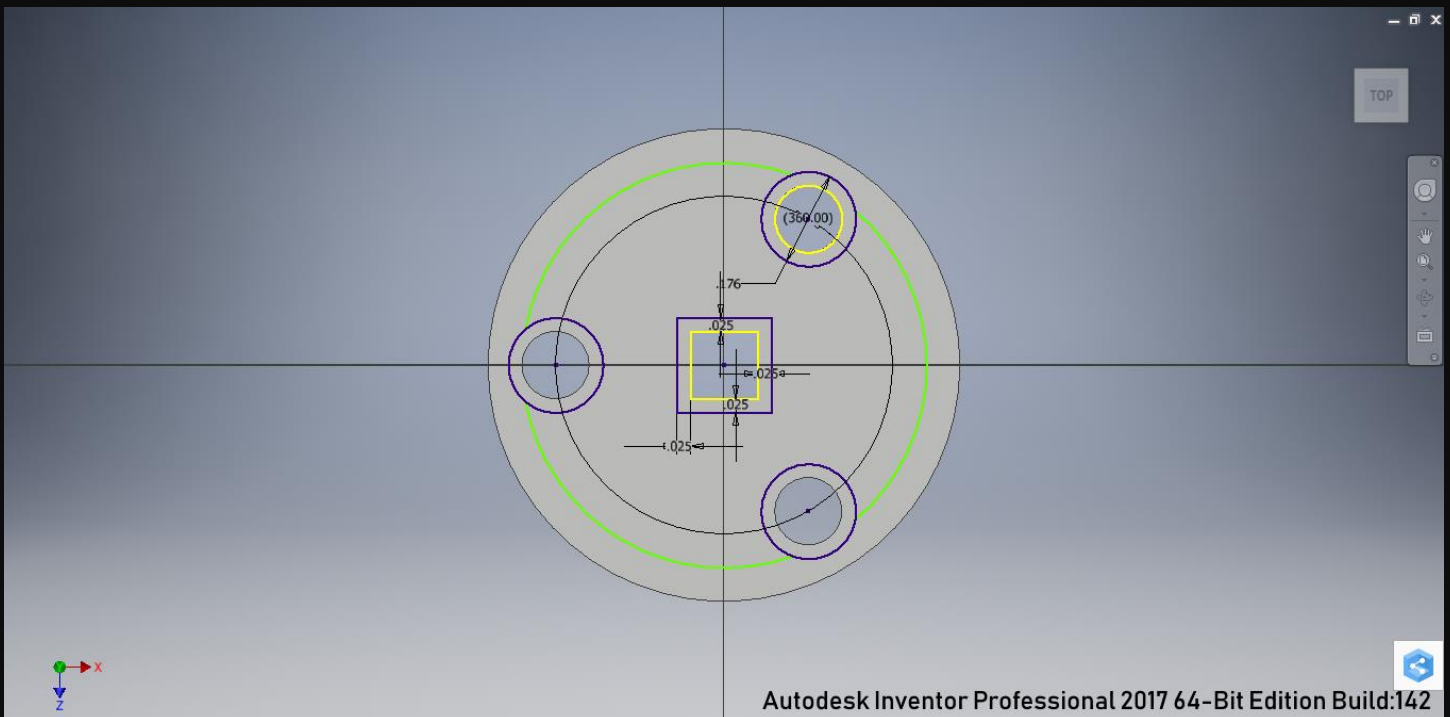
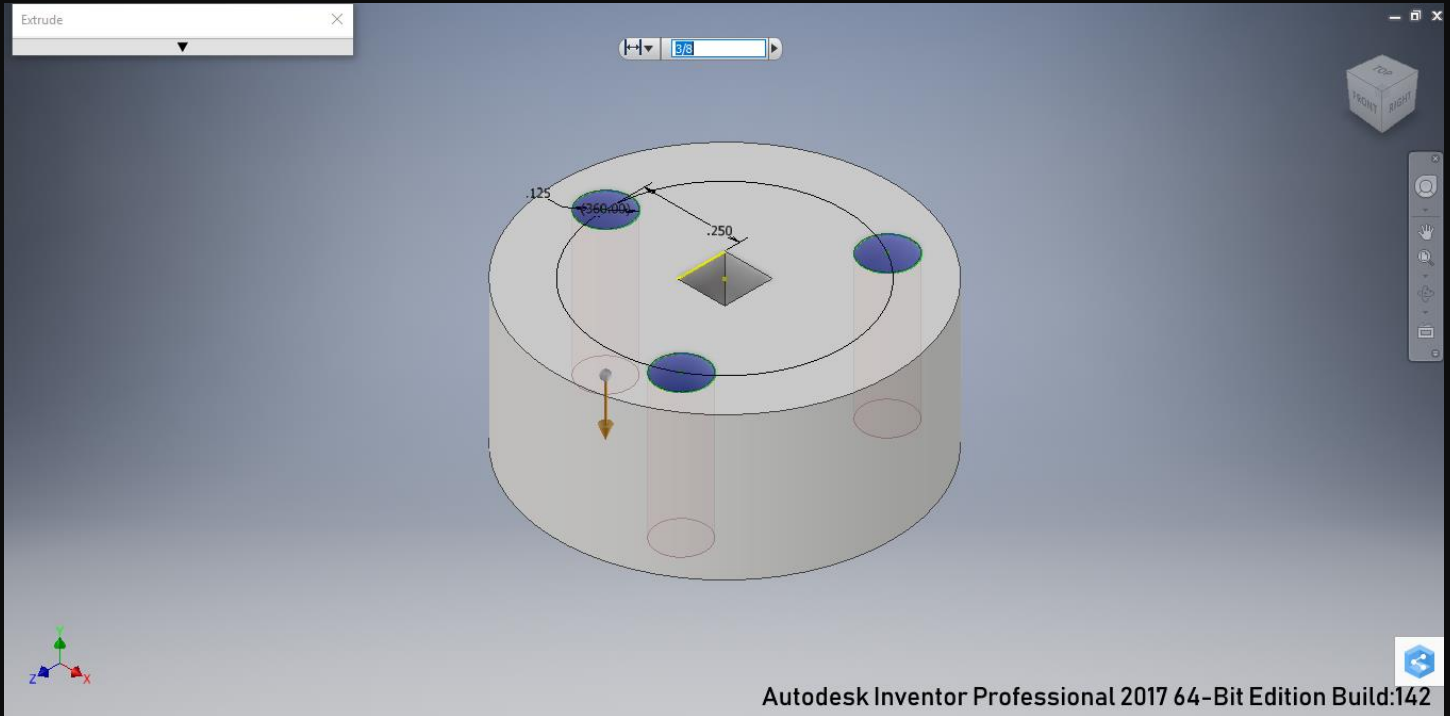
Then, a hole for inserting the axle was created in the center of the cylinder's top face with the rectangular sketch tool and cut with the extrude tool.



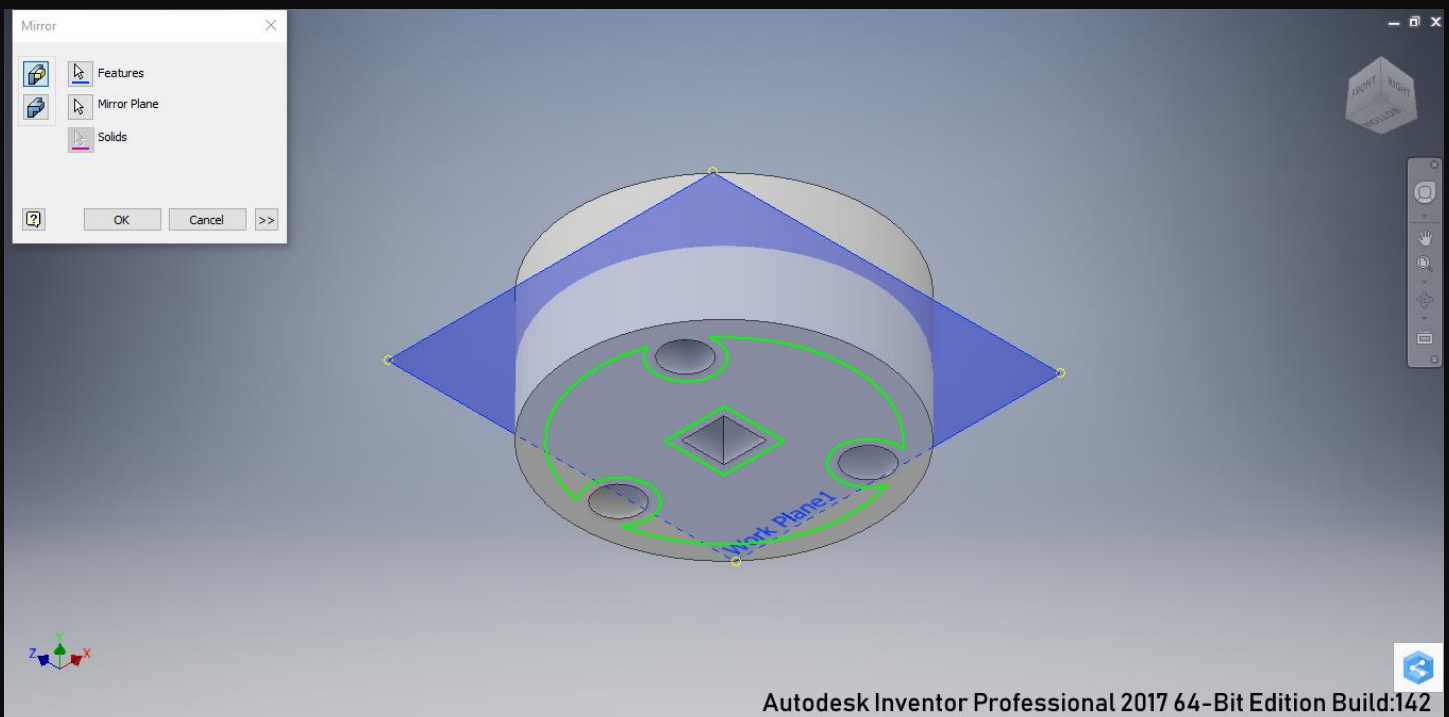
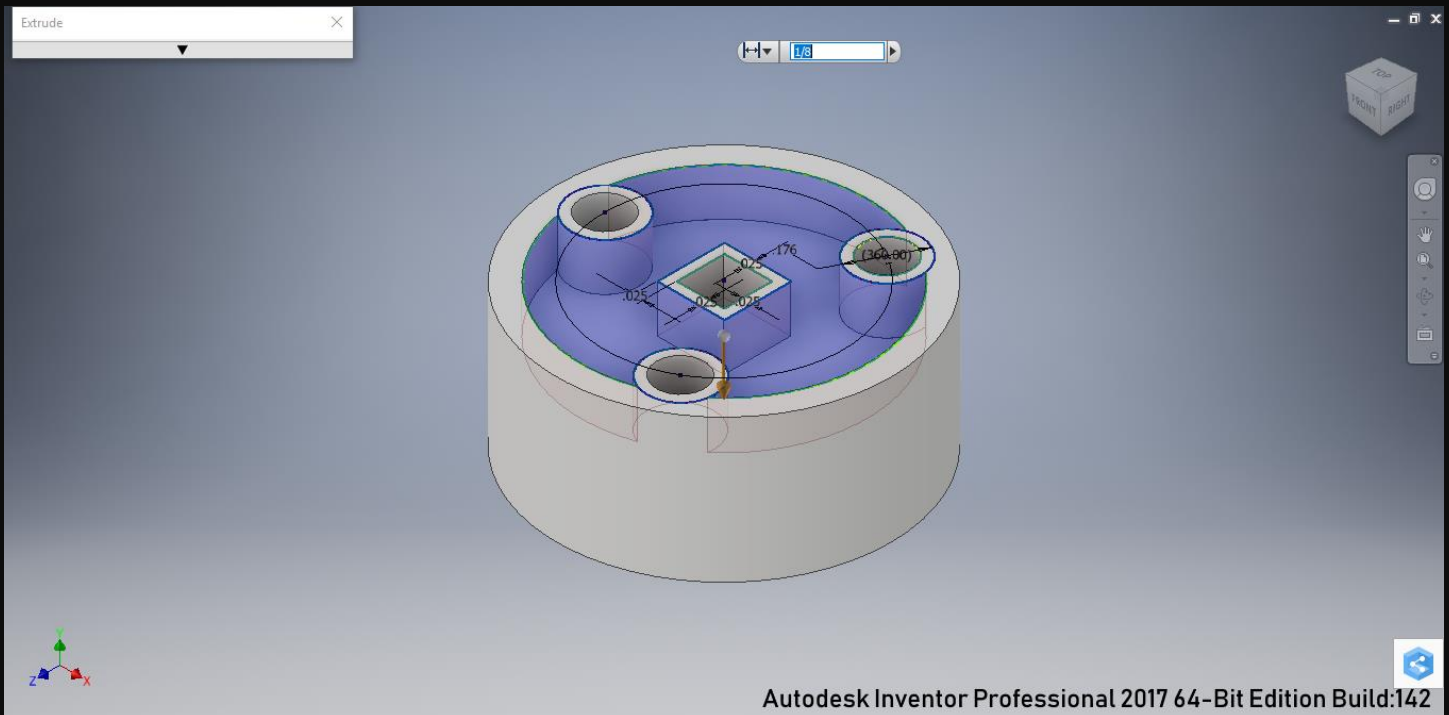
To create the holes for the rods, a circle sketch was created above the hole for the axle, then repeated around the cylinder's top face three times with the circular mirror tool.



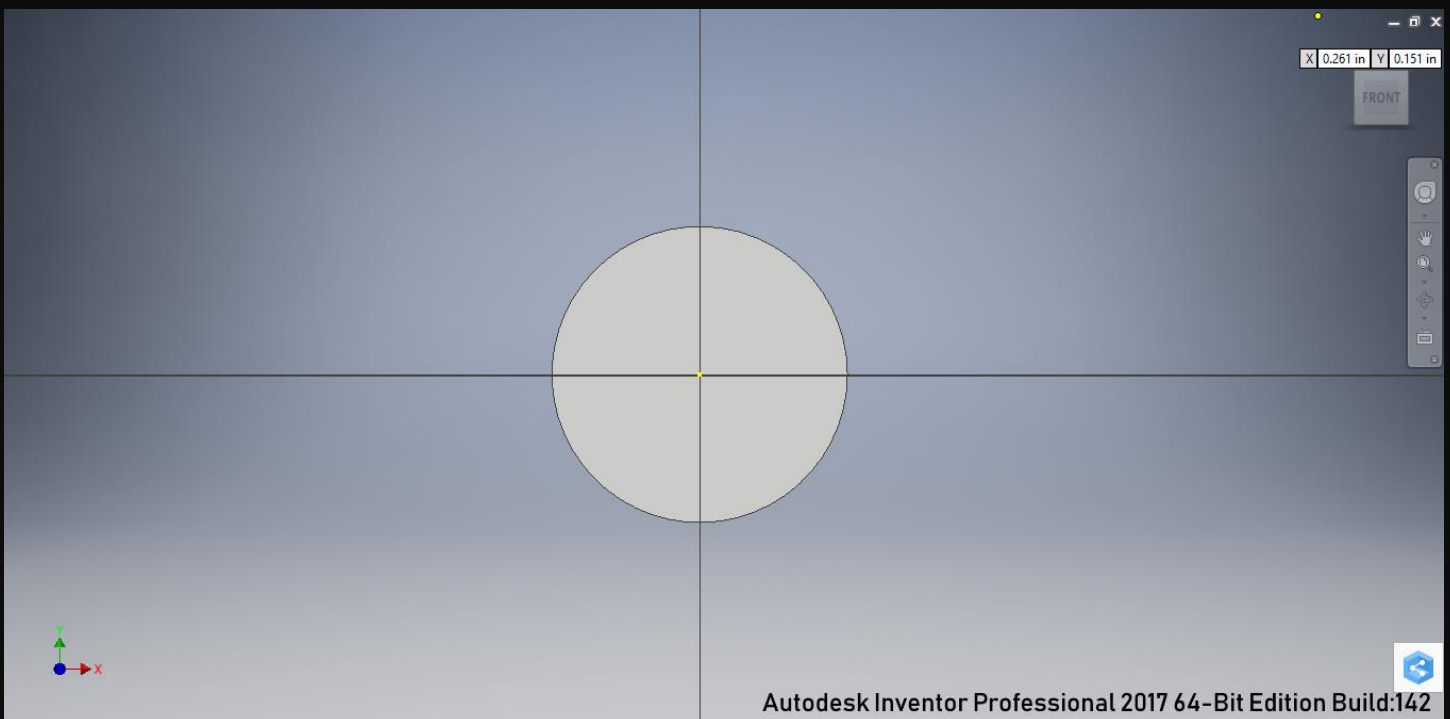
The holes were then cut through the length of the cylinder. The features of the face of the cylinder were outlined in a sketch using the circular, linear and trim tools.



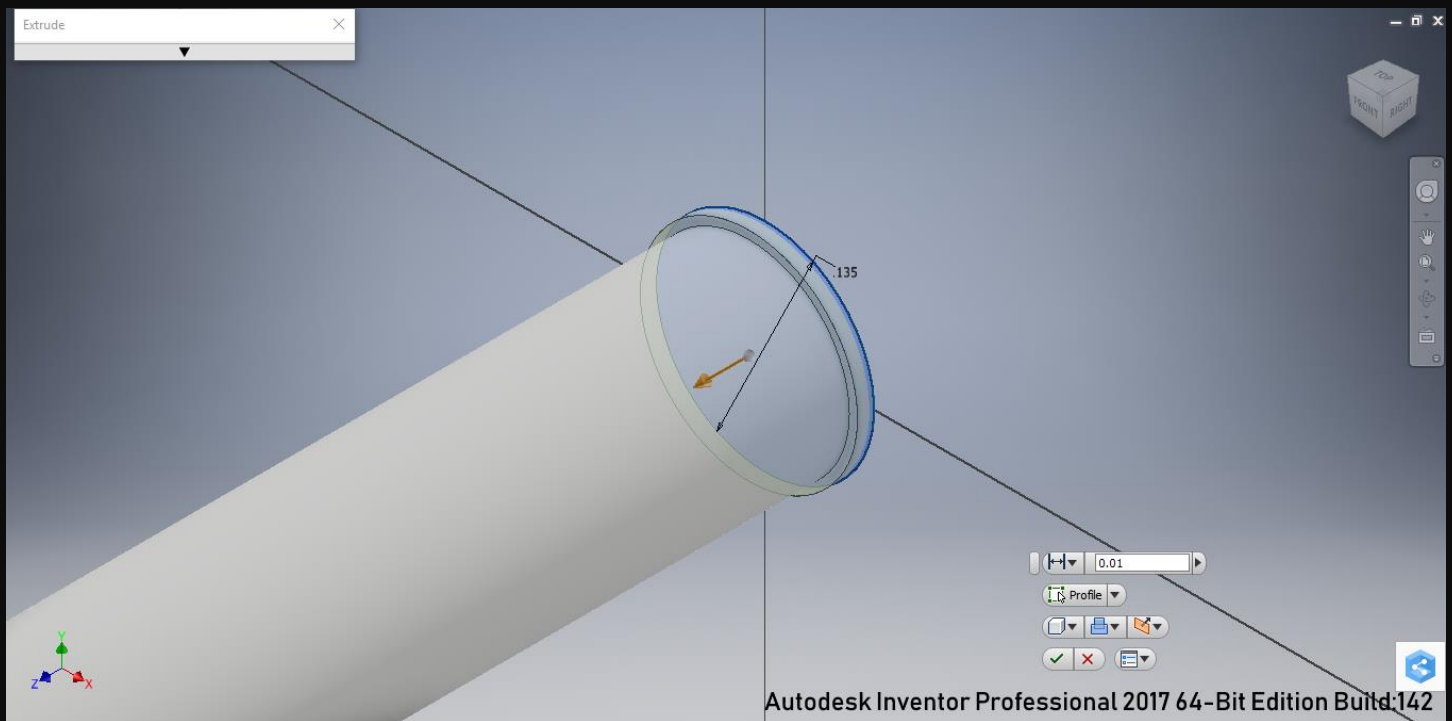
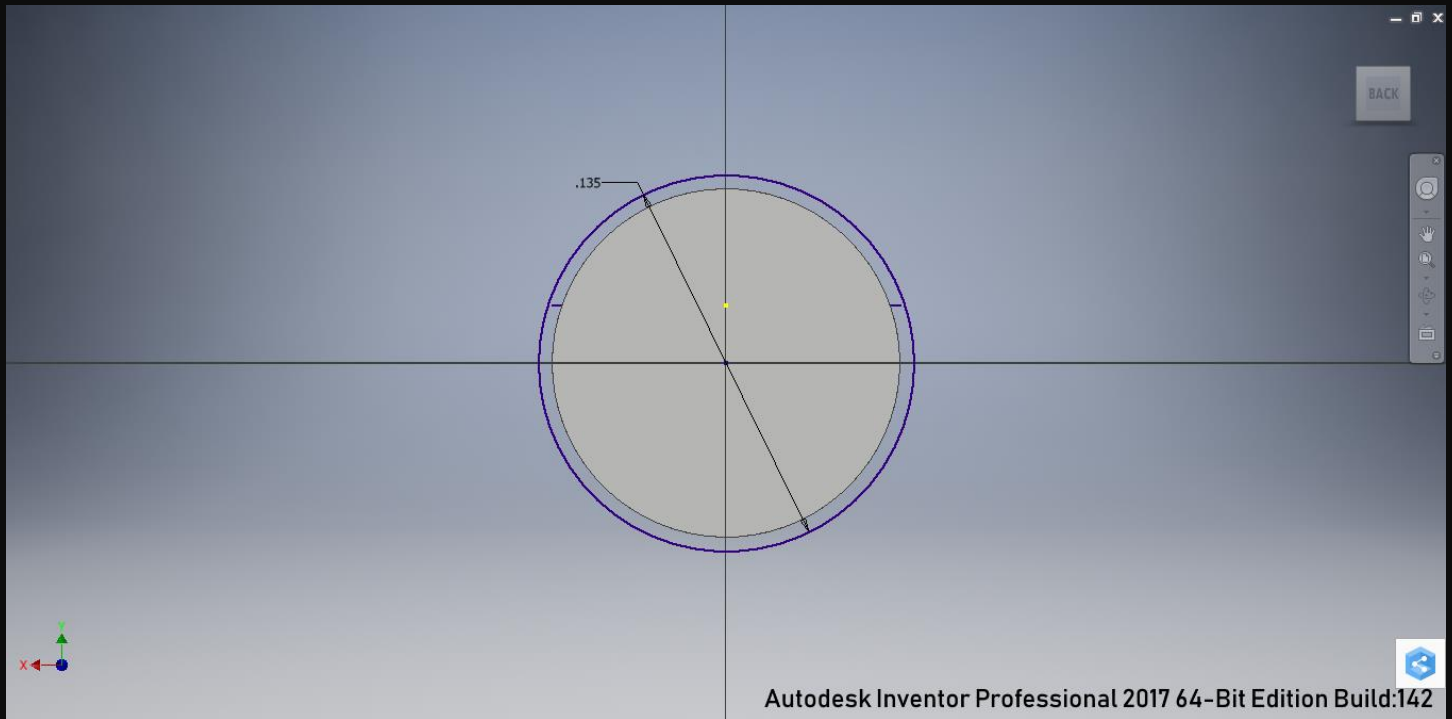
The sketch was cut part-way into the cylinder, then the plane tool was used to create a plane in the middle of the part to mirror across.



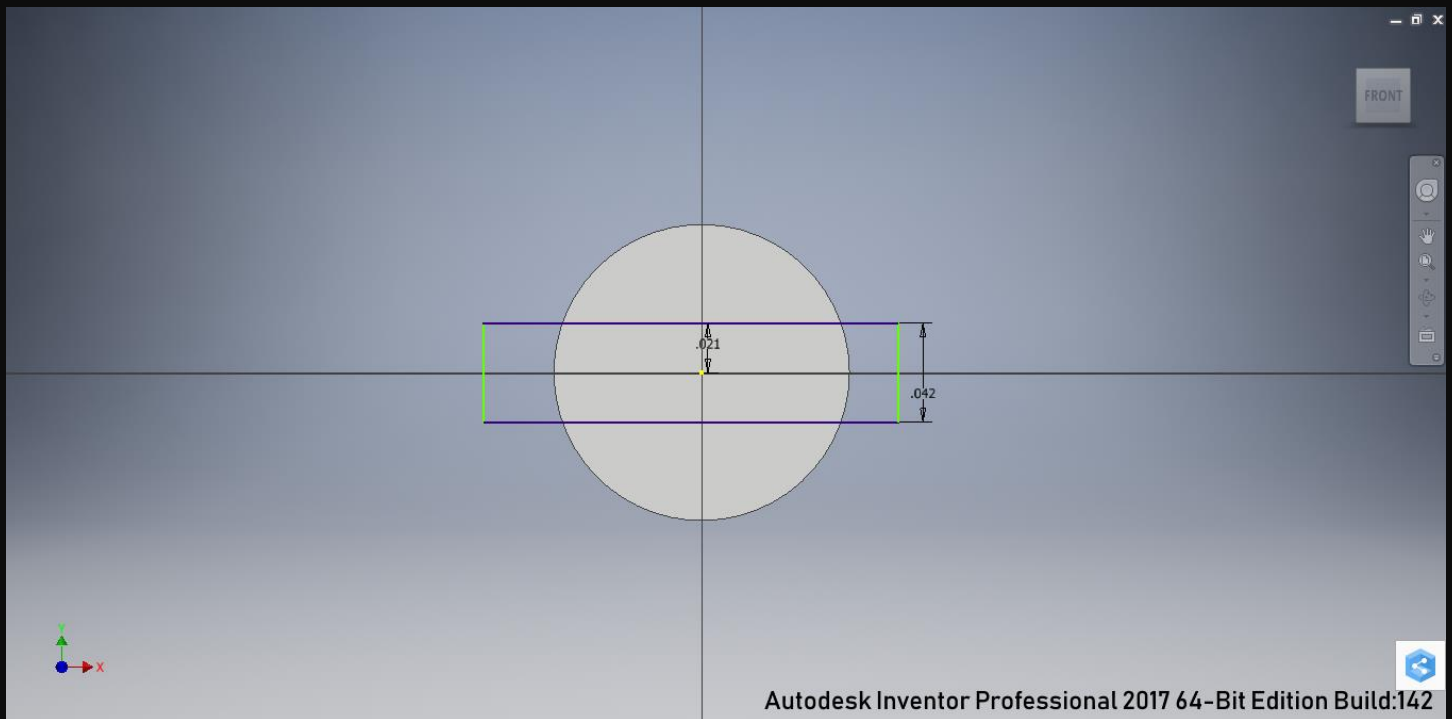
After this, the rods were created repeating the step of creating a cylinder and extruding it to the desired length. Then, a sketch was created on one end of the rod.



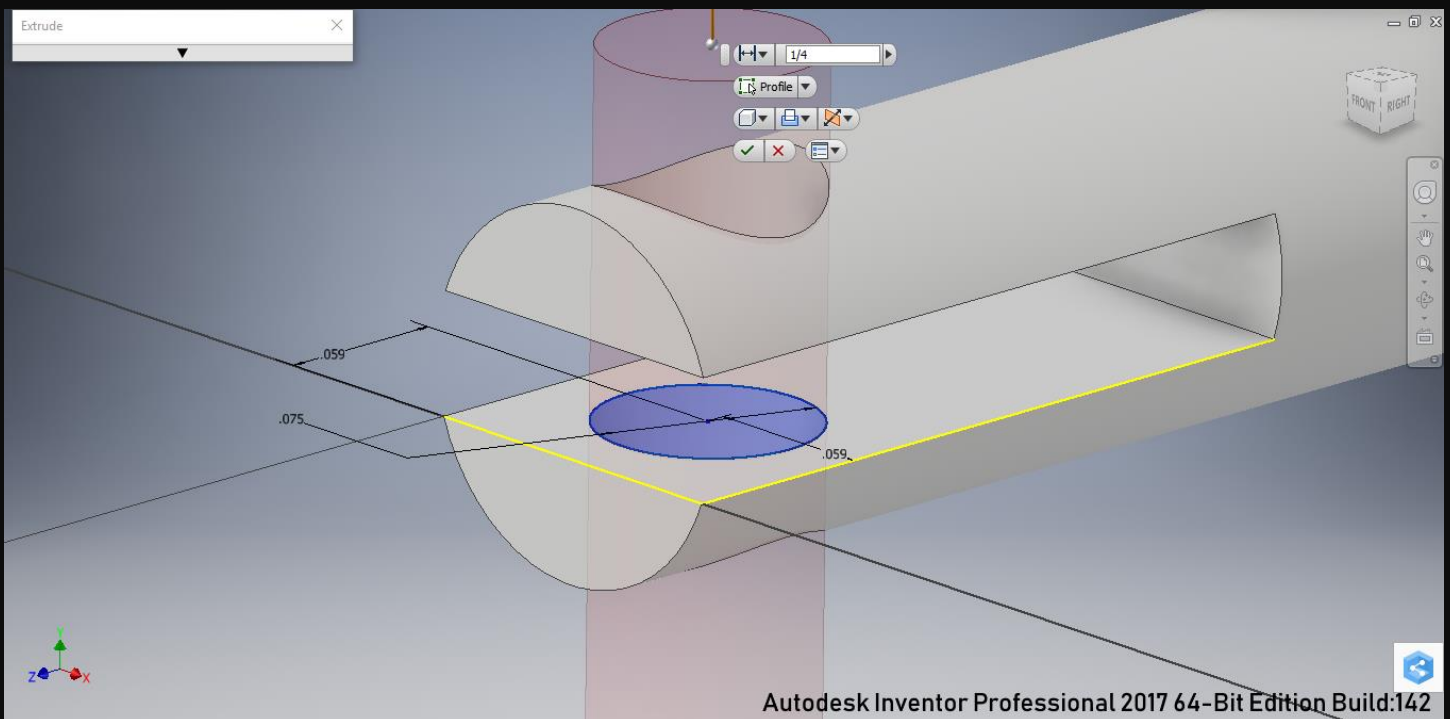
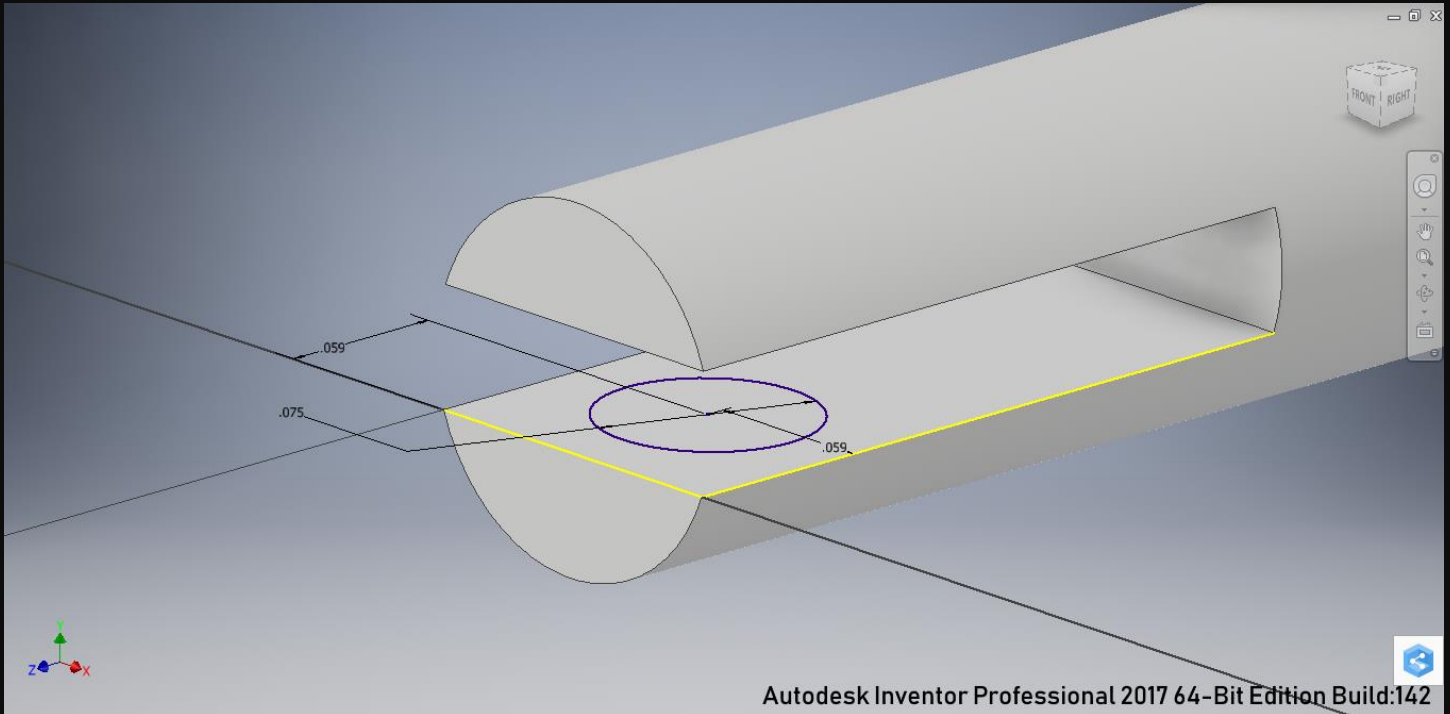
A circular sketch was then created that's slightly larger than the rod. This was extruded a short distance into the rod, creating a thicker end that will stop the rod from sliding out of its holder.



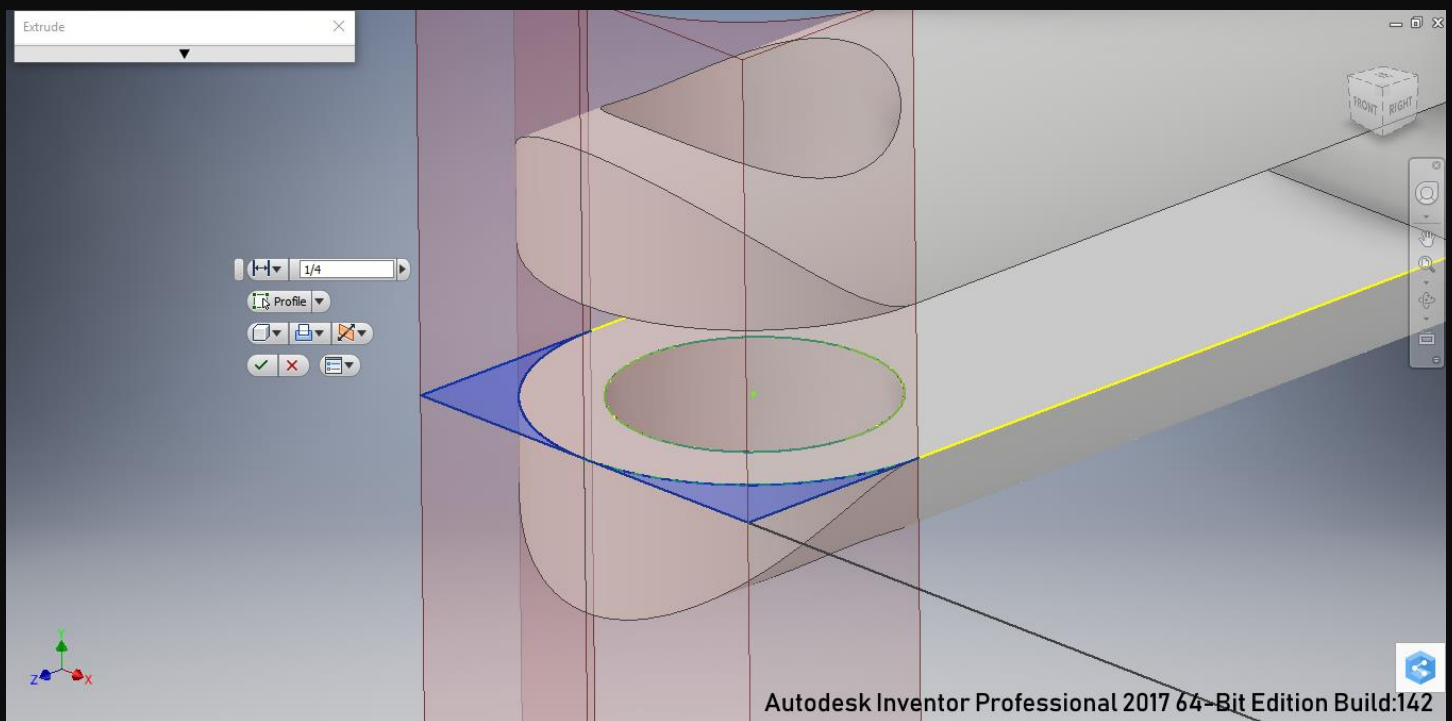
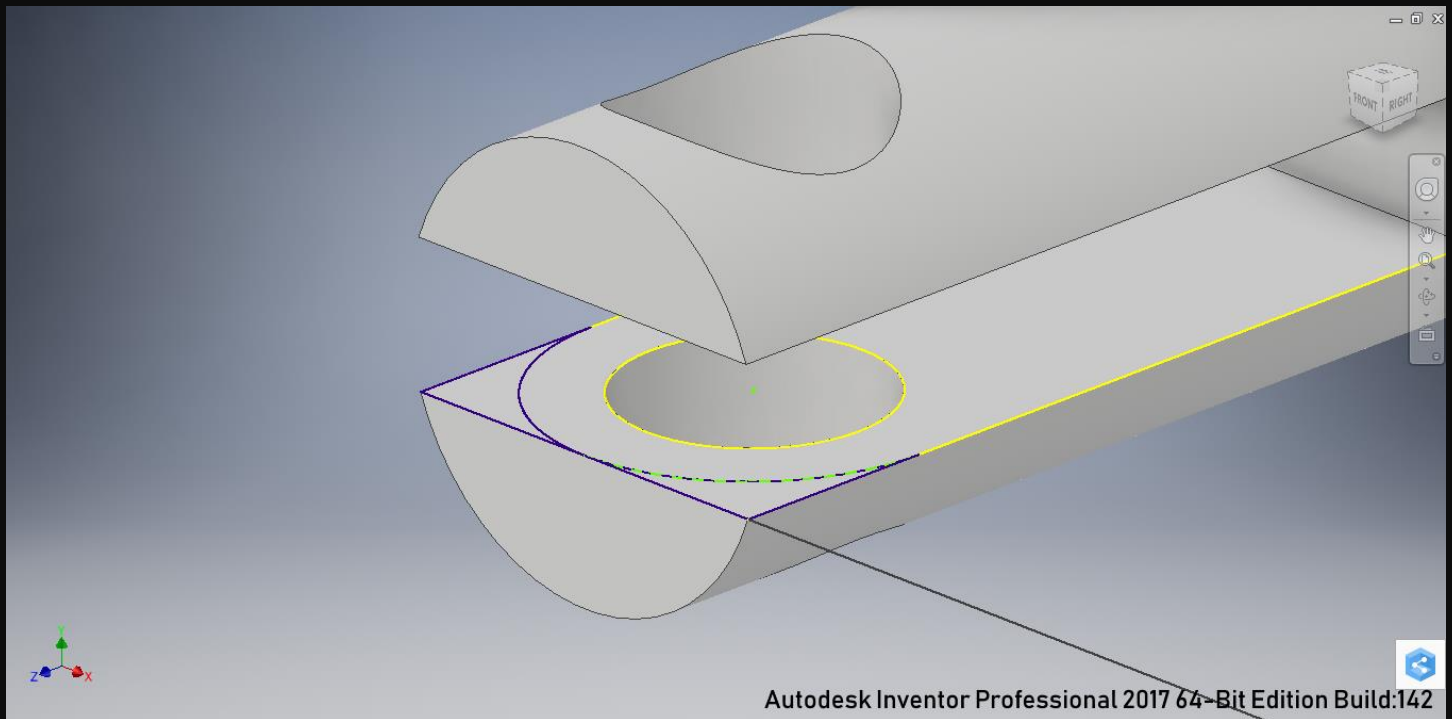
After this, a sketch was started on one of the end faces. From this point, the rod would be made either into a type A rod, or a type B rod. For type A rods, a rectangular sketch about 1/3 the height of the cylinder's face was created at the face's center. The sketch was used to cut a notch into the rod that the type B rod will later fit into.



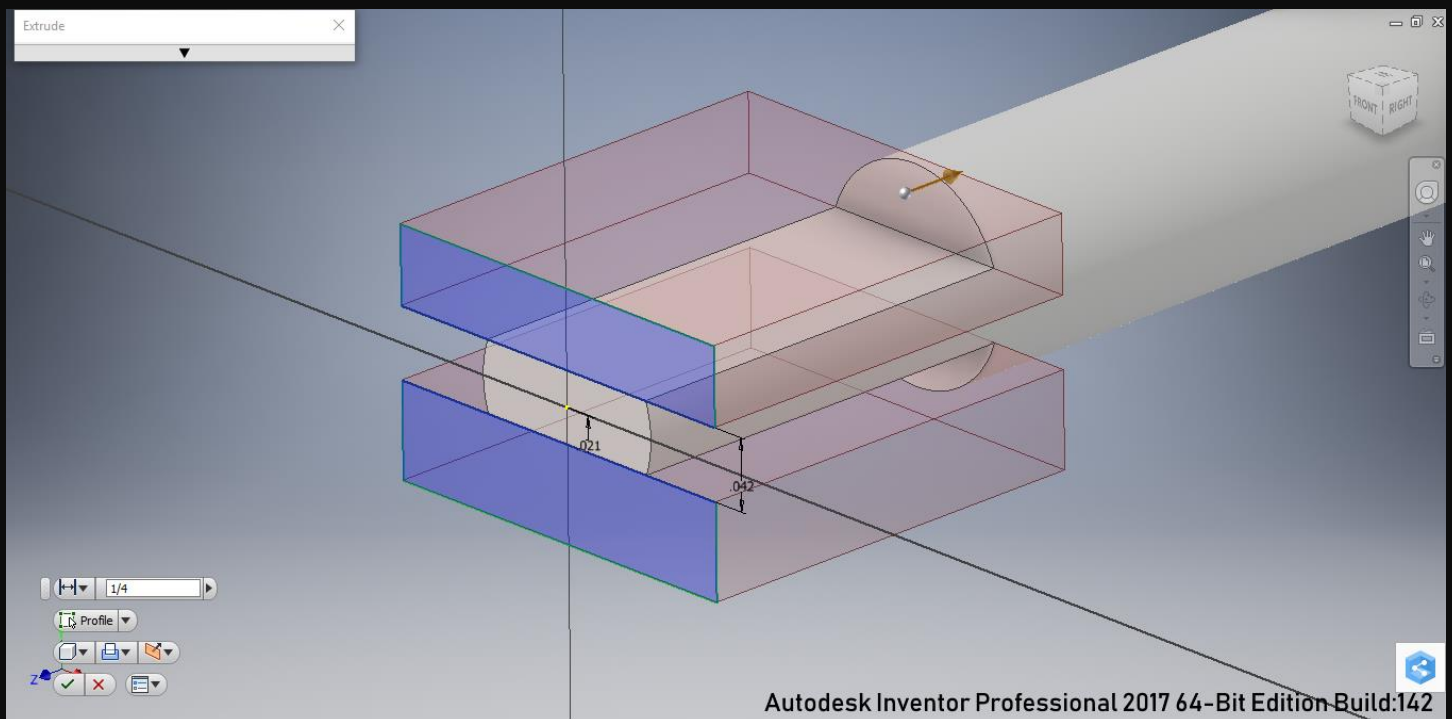
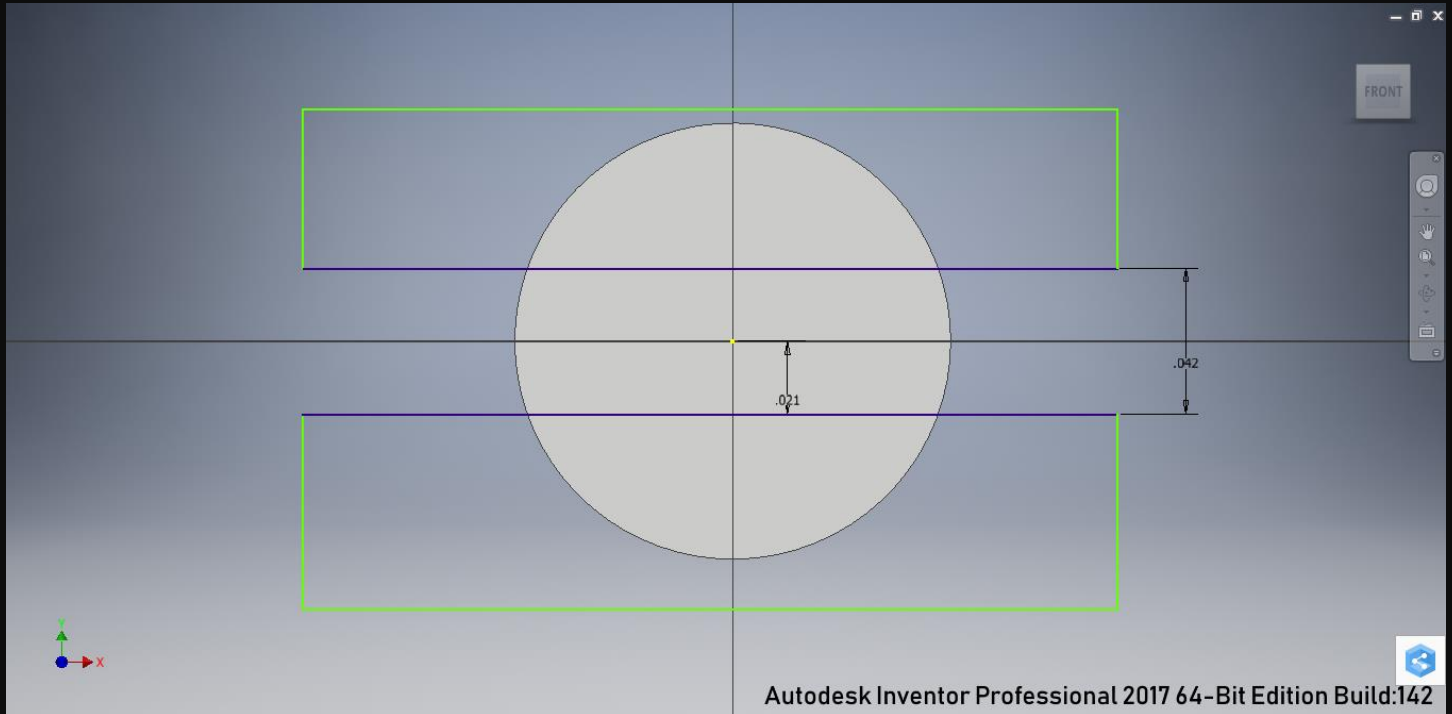
A circular sketch was created inside a notch, then symmetrically cut to create the holes for the pin that will hold the rods together and allow them to rotate.



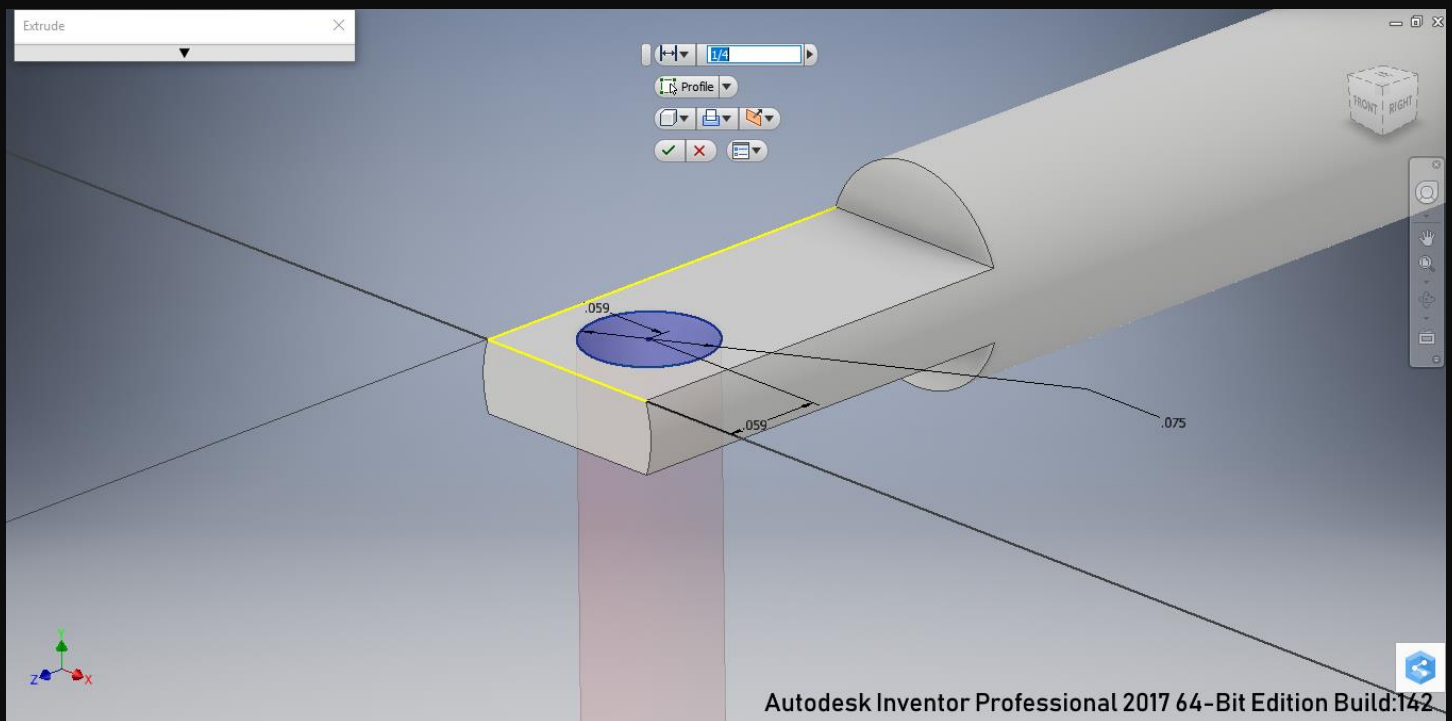
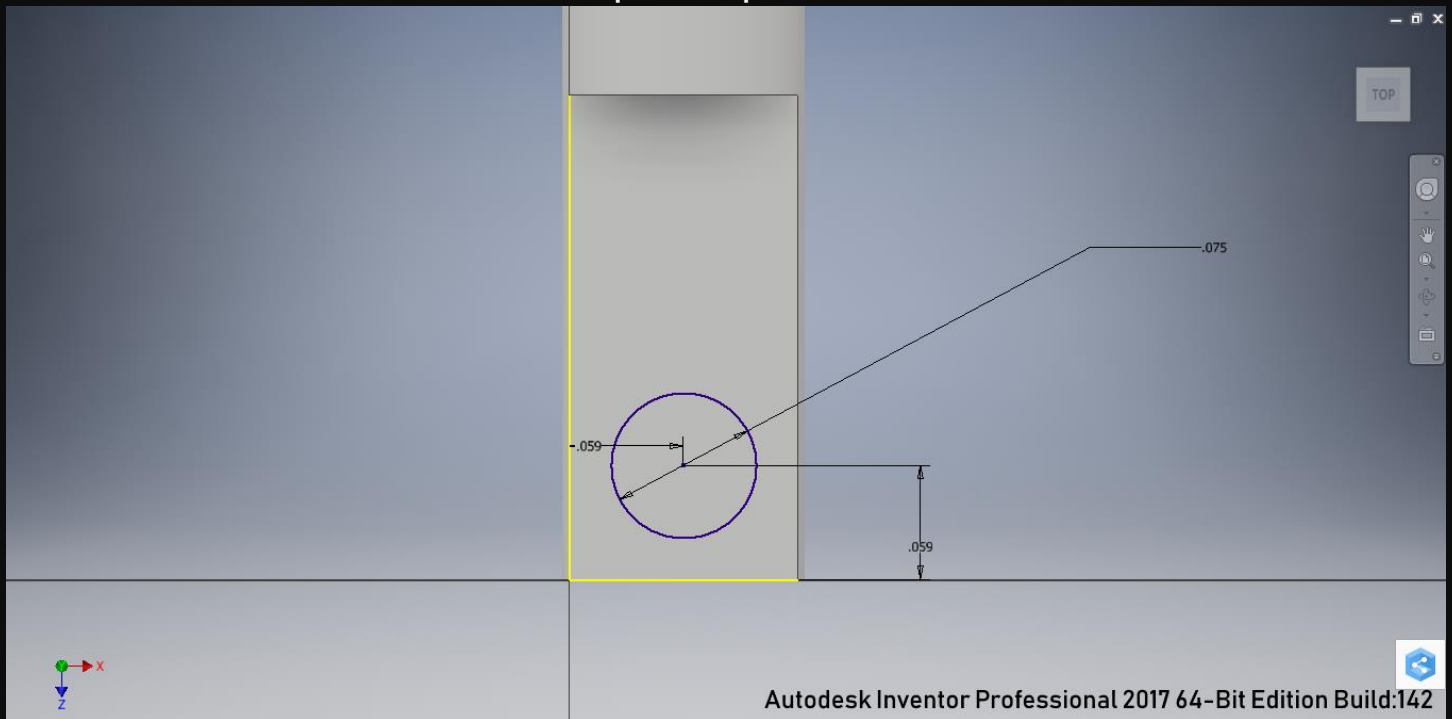
The end is then rounded off using the linear and angular sketch creation tools.



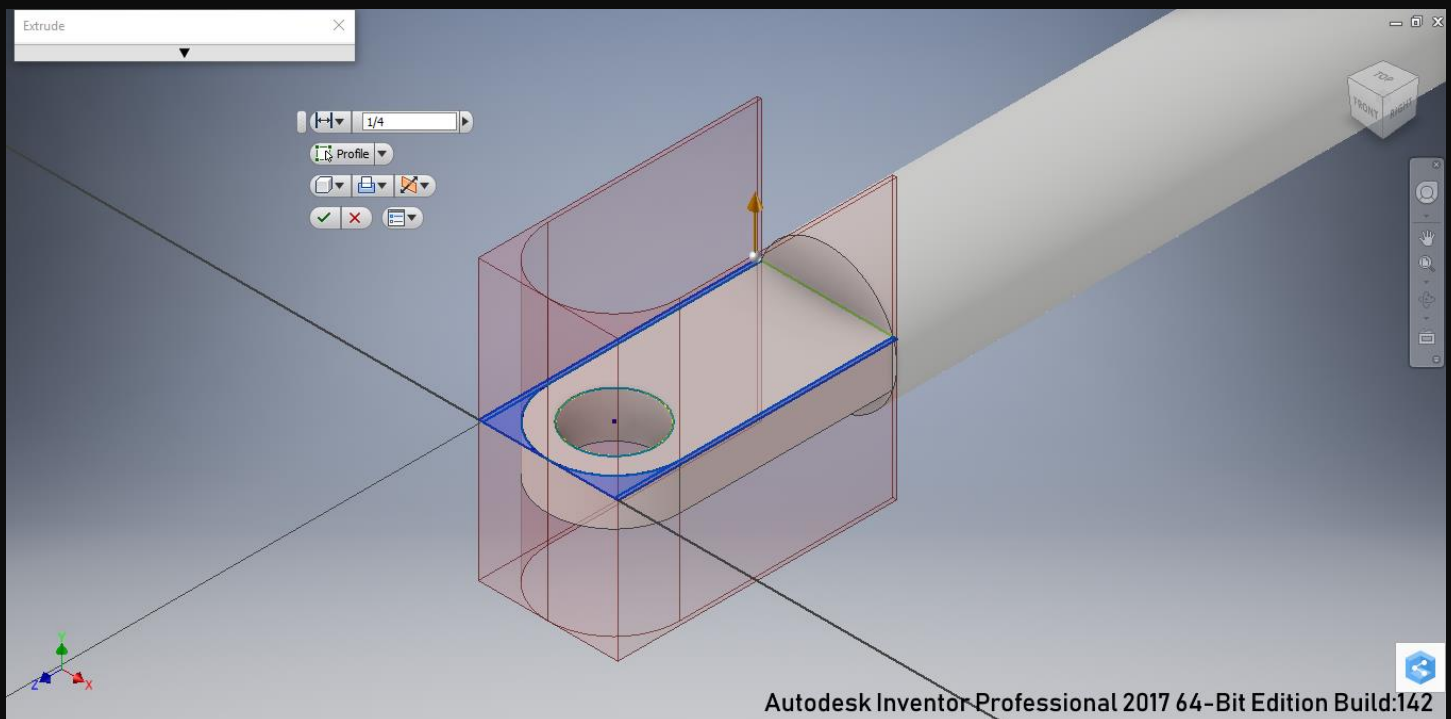
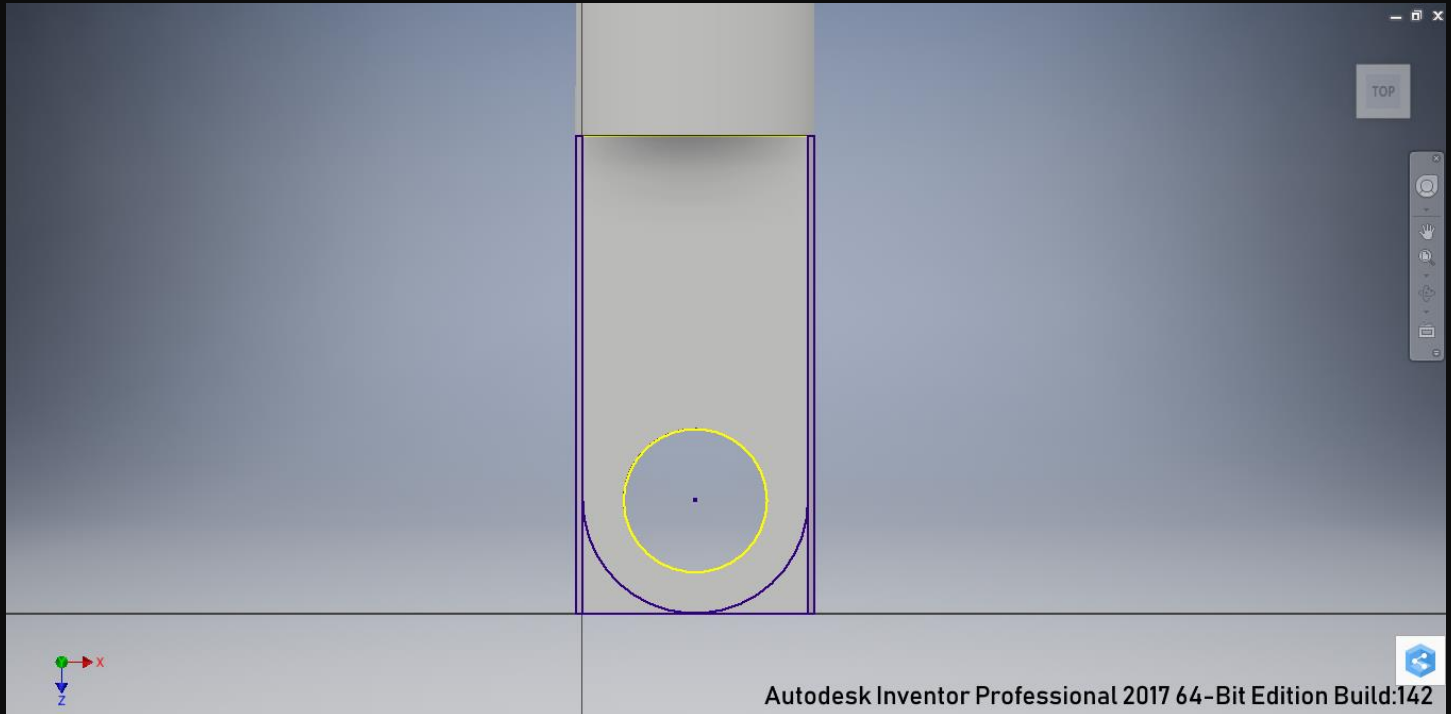
For the type B rods, the inverse is done, all of an end face is covered with a sketch except for the middle third and then cut out.



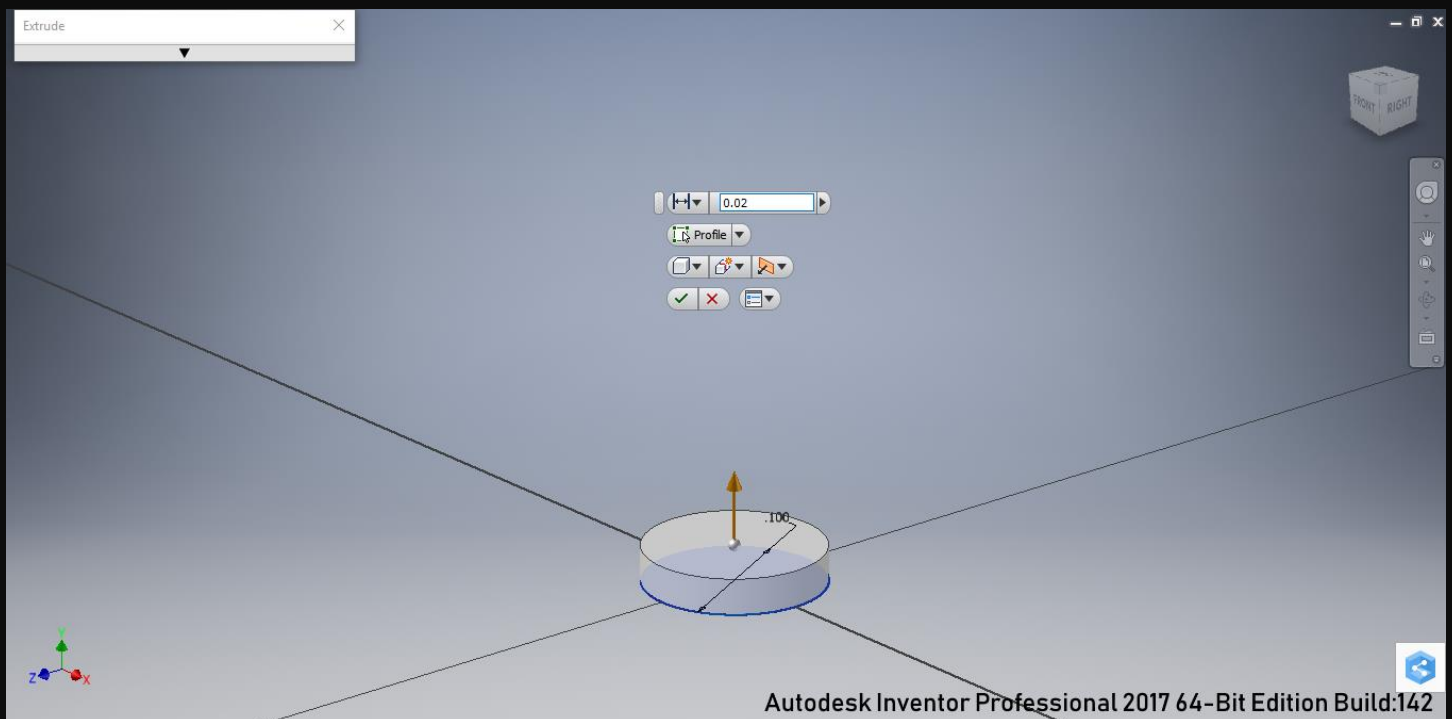
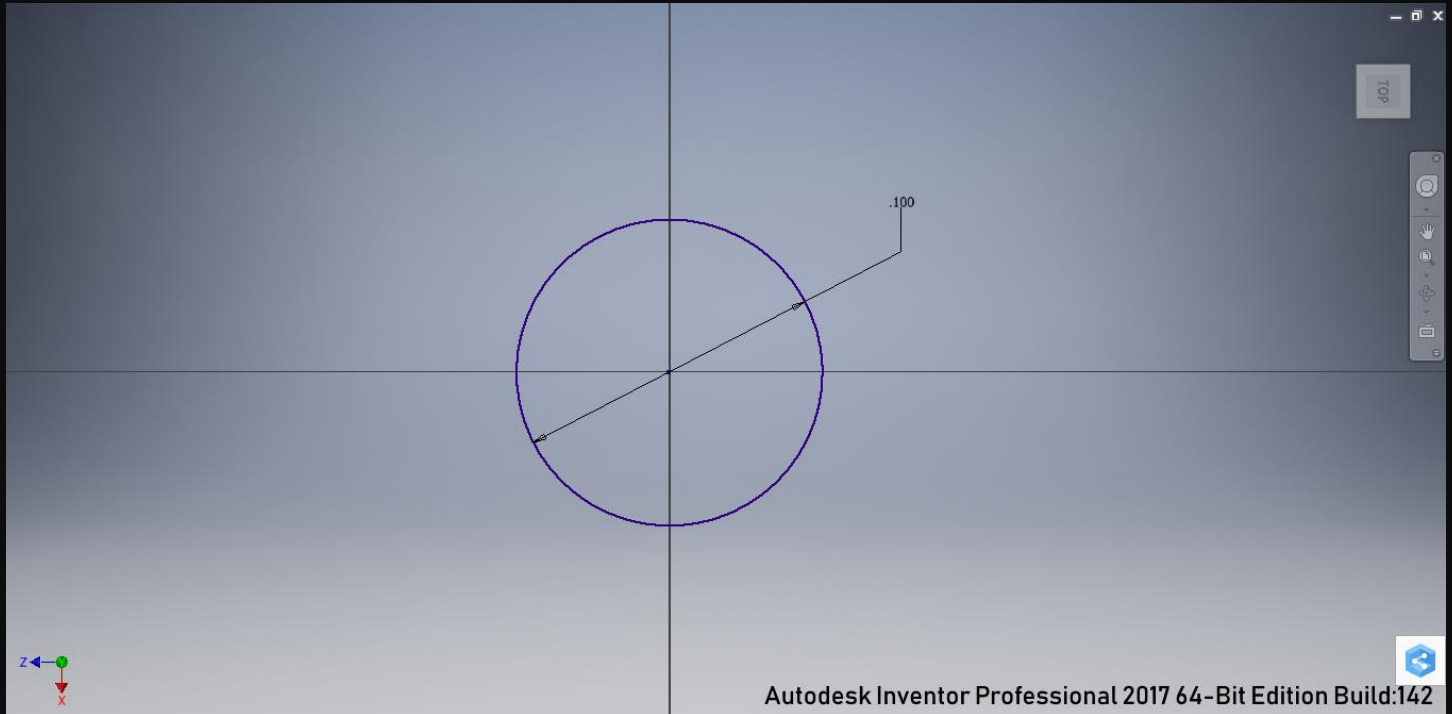
The method to create the hole for the pin is repeated here.



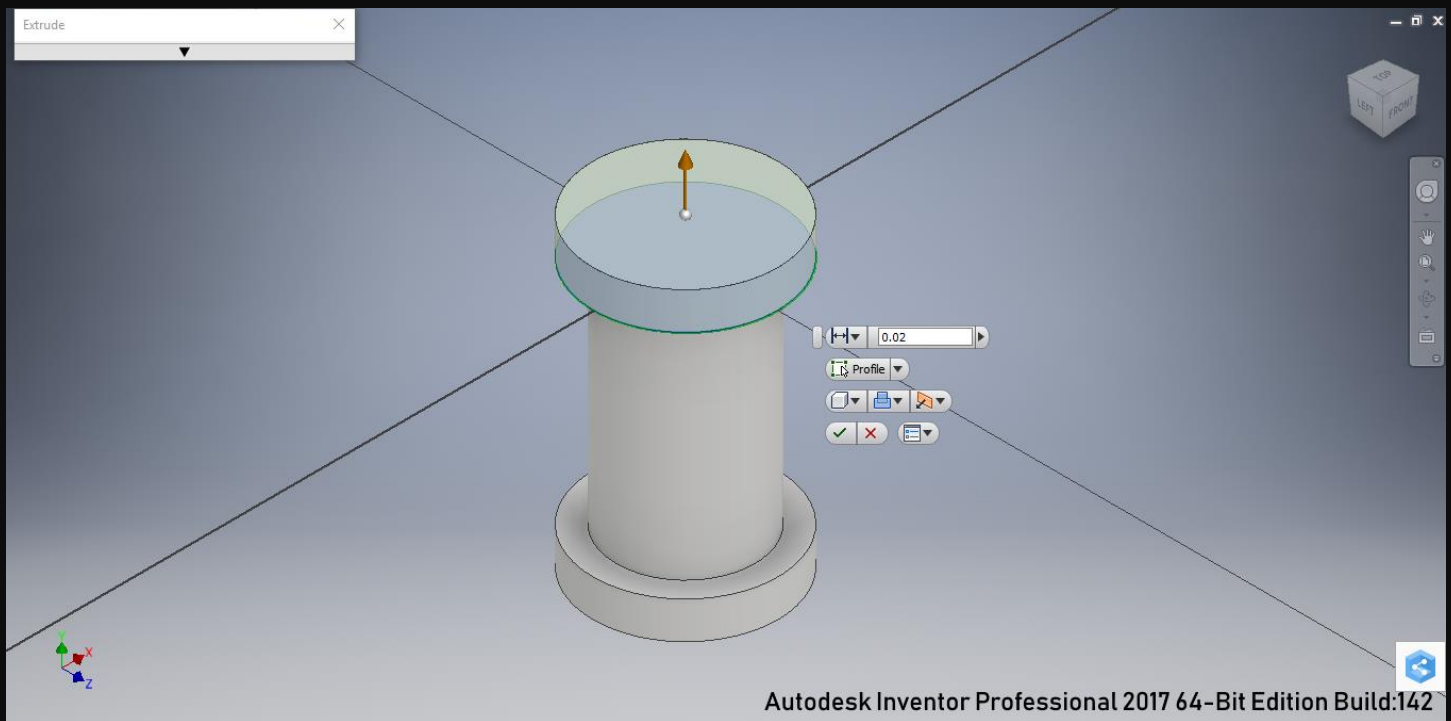
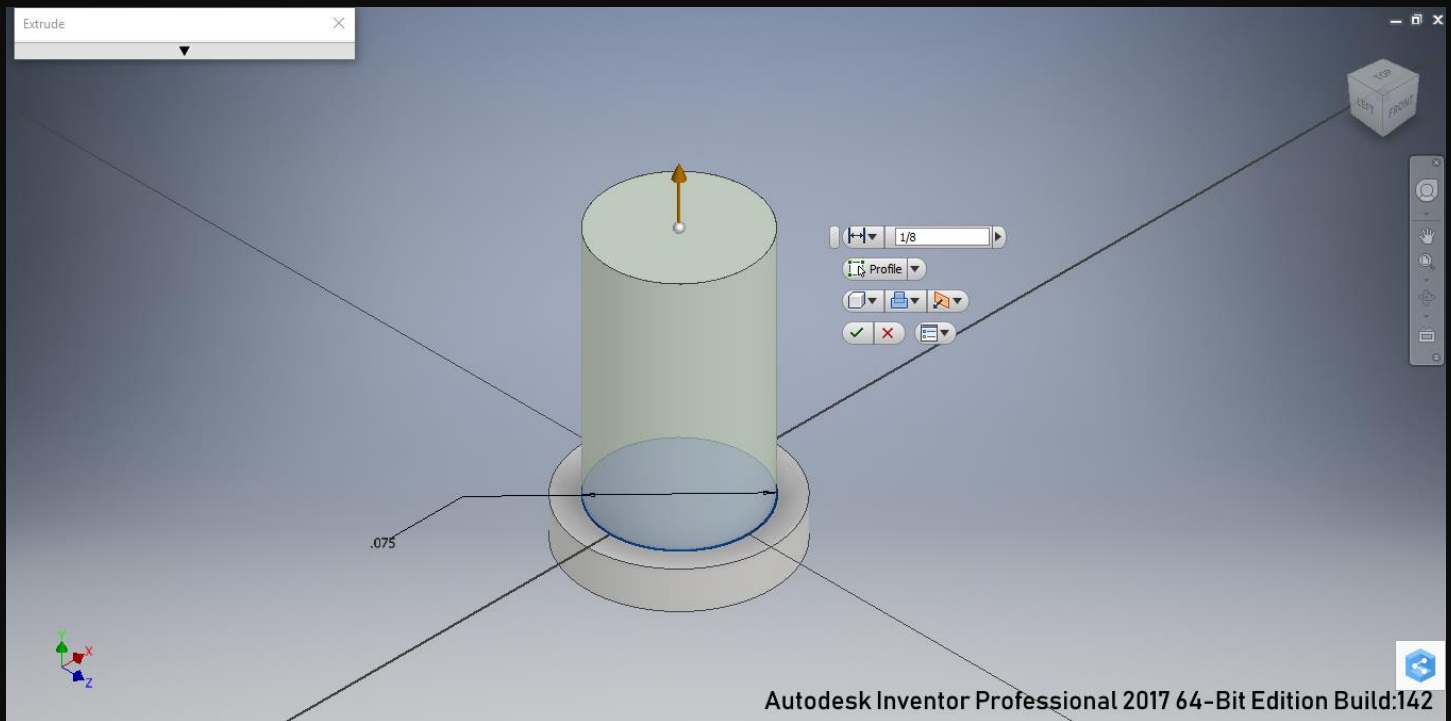
The method to rounding off the end is repeated here, though due to the rod widening out, rectangular sketches are created on each end to trim it to the width of type A rods.



To create the pin, a small circular disc was created with the circular sketch and extrude tool.



After this, another smaller circular sketch was created and extruded to be the same height as the rod's width. Another circular disc is created at the end of the cylinder in the same way as the first part to create the pin.



Due to the part's size and complexity, there's a larger "3D-printer" option for the parts that have been adjusted to be printed for creating demo models.



Regular (7/8" Diameter)



3D-Printer Friendly (2" Diameter)

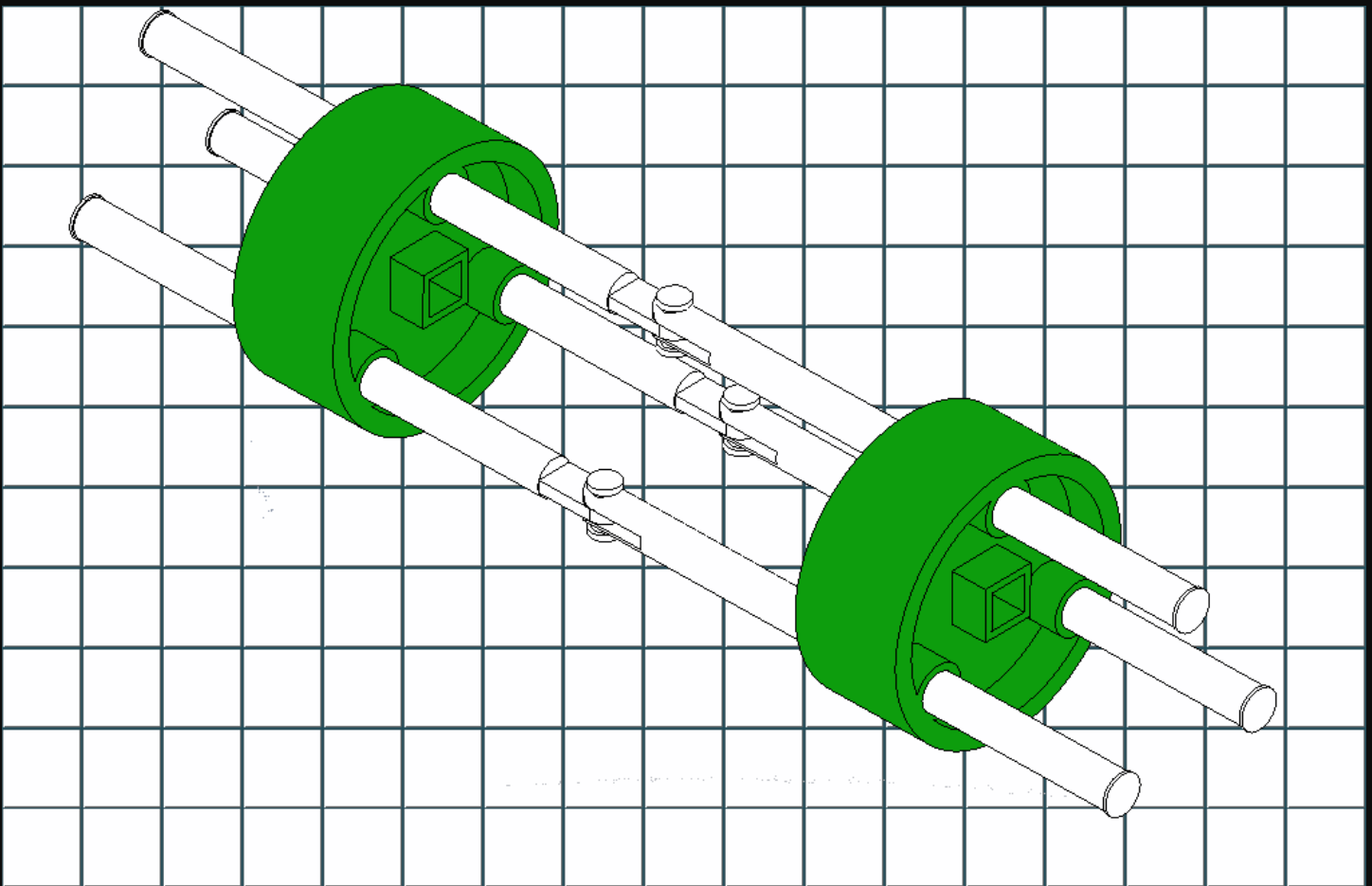
Software used: Autodesk Inventor Professional 2017 64-Bit Edition Build: 142



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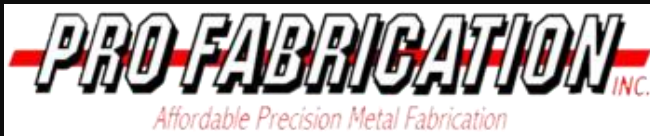
Conclusion

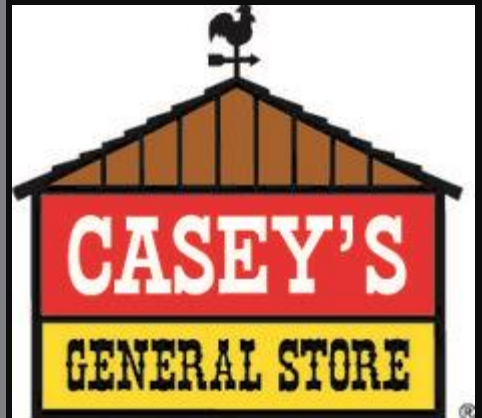
The creation of this part taught me a great deal about the engineering process behind the creation of a part to solve a problem. I learned about having to pursue multiple problems to research to find which one would benefit the most from a solution, and what the most efficient solution is. The creation of this part also gave me more experience in using Autodesk Inventor, as well as the full process between thinking of a part, creating a virtual model of it then 3D-printing it for use. I expect to be using this program more in the future for creating models while creating products. Proper usage of this software is very valuable for a team hoping to compete at higher levels due to it being useful for creating accurate 3D representations that could be used in engineering notebooks or presented during judging.



For my future career path, Information Technology, this software can still be useful. This software is very useful in creating detailed designs for products, including computer products. Alternatively, it could be used to render designs for use on websites or professional product brochures. Having experience in 3D modelling is overall a very desirable skill in the increasingly digital age.

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