

2414X Make It Real Challenge Submission: High Strength 12T Gear Circular Insert



From Spring Valley, NV, USA

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What's The Current Problem?

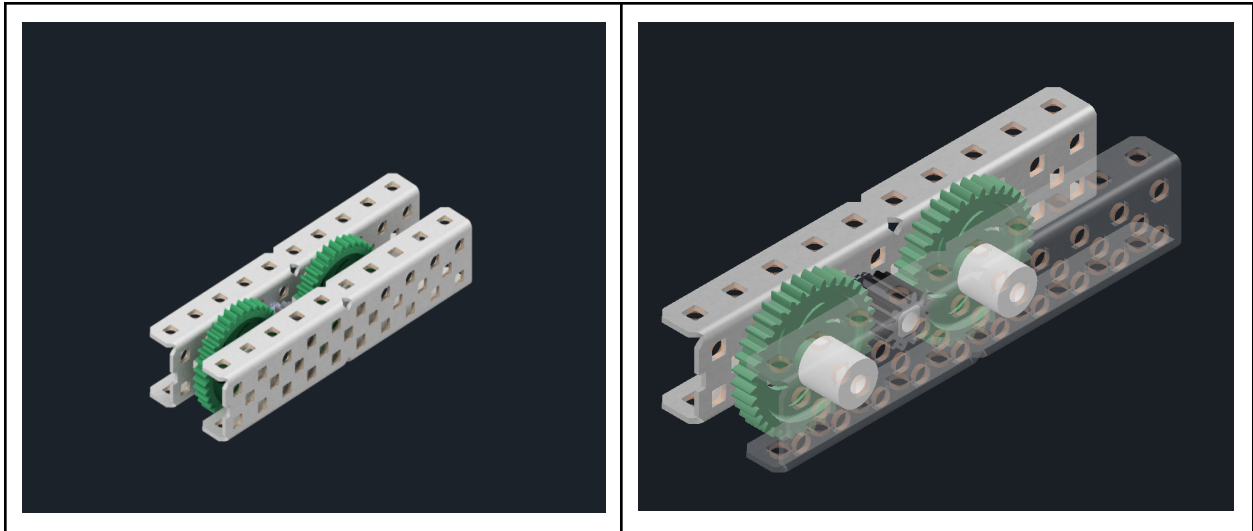
Currently, the 12 tooth gears sold by VEX Robotics do not currently have an option for circular inserts. At many higher levels mechanically, "dead-shafts" are used, which are shafts in which they don't spin but constrain the spinning object axially. These are often called screw joints in the context of VEX. For any shaft unpowered it has become a preference to have it be a dead-shaft as to having a live-shaft, as having a live-shaft is unnecessary amounts of friction as the part is constantly rotating which is constant contact. This is as the shaft has to rotate through the bearing and when anything is contacting another part it causes friction vs it being stationary, but the bearing aligns it axially. Screw joints also act as if you're to have a screw running across both sides offering additional support to the c channels the joint is mounted to.



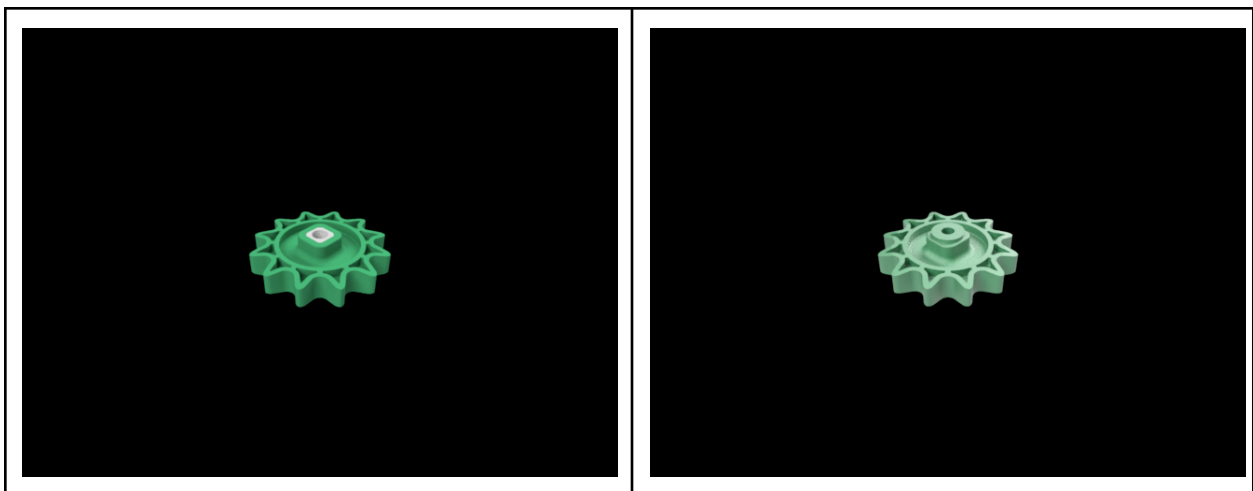
7700R's Tower Takeover Anti-Tips on "Screw-Joints".

Possible Usecases for this part?

- Idler Gear In A Gear Train of 3+ gears



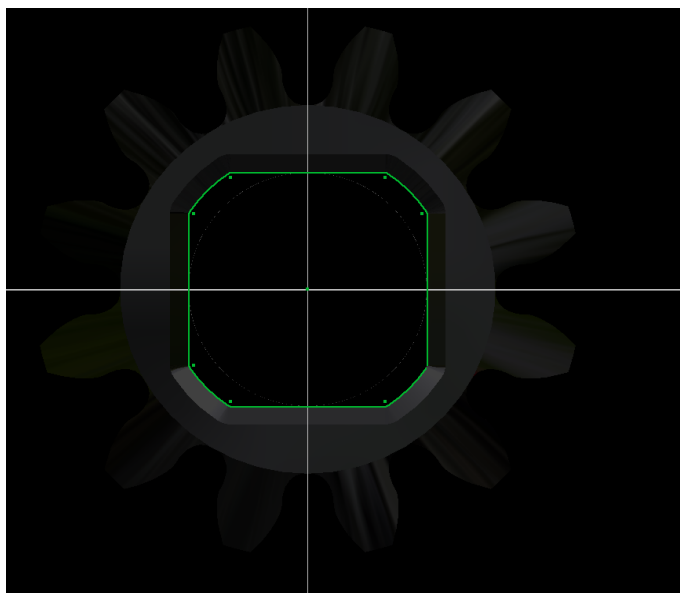
- Insert for HS Sprockets to allow flush free-spinning vs the added 1/8th



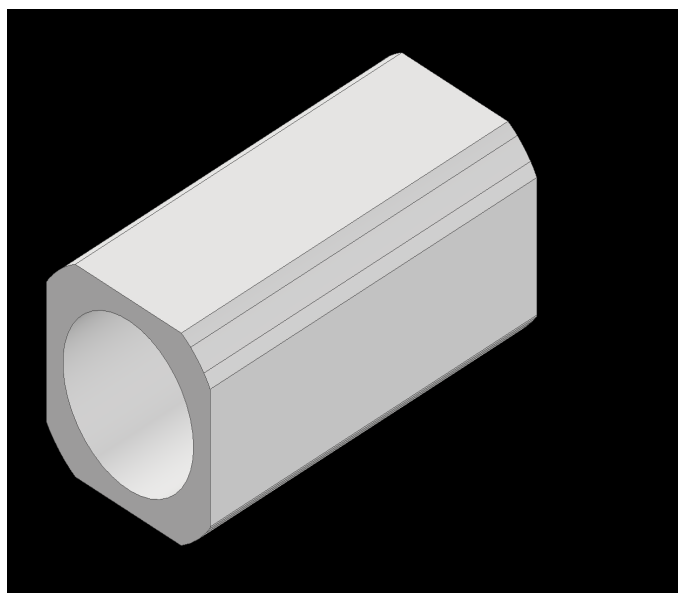
The Part: How and Whys?

This part was designed in the desktop application Autodesk Inventor 2022, To start this I first opened the file for the original 12 teeth high strength gear with no insert in the assembly. The reason for opening it in an assembly is as you can then create a part based on the design of another part. I then select the face of the gear, then start a sketch based on it. I use the project geometry tool to get the correct geometry for the insert to then be able to sketch the rest based on that too. I then just needed to add the screw hole which was assumed at .2 vs .189 of which it actually is as I must design around the printer tolerances and assume it shrinks. I then extruded it .5 inches which will become the thickness of the gear. I finally had a part that would work but some things had to be changed to make it more user-friendly as inserting it would be very difficult and require a considerable amount of force. Filleting is the act of rounding something in mechanical situations, inventors Filleting tool was used for my part. I then filleted the edges .025 inches as it was the biggest fillet possible besides .026 which is a more arbitrary number so I chose .025 for the sake of memorability and ease. I then tried this again without using these tools and found that manually sketching was a significantly slower process. The project geometry tool made it so I didn't have to constantly use the measure tool while sketching.

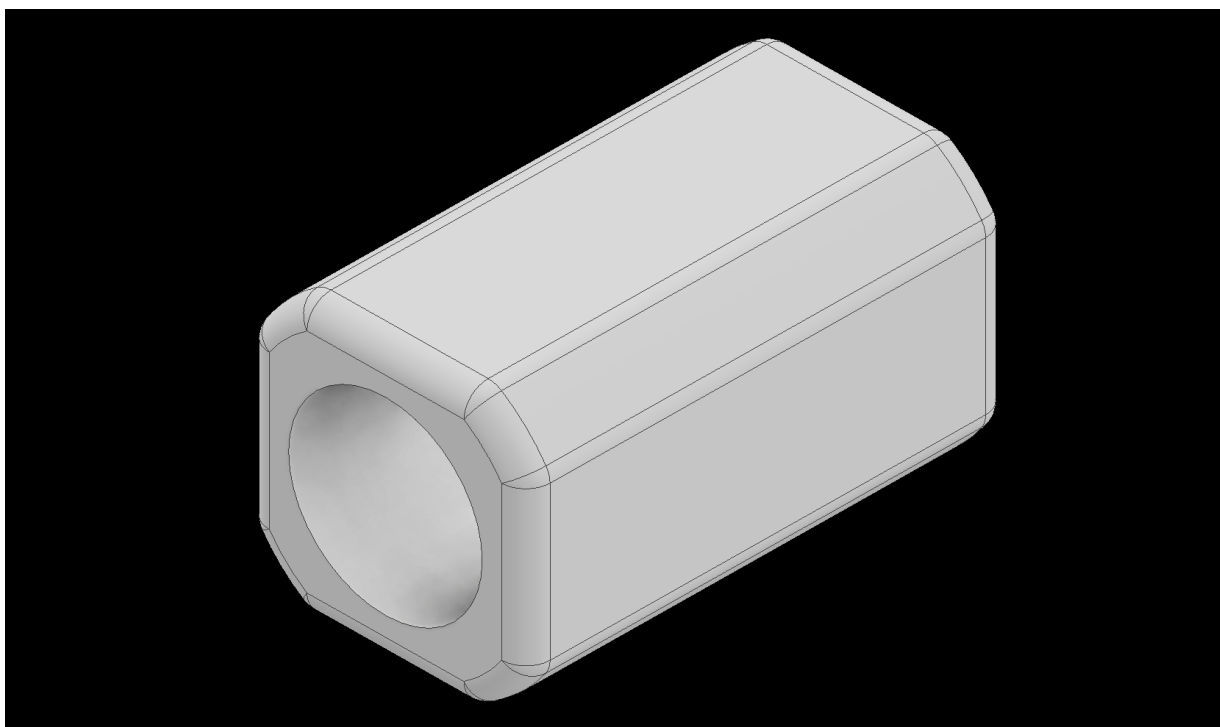
Images of the part



The Geometry Projected onto the sketch hole.



The part extruded with the screw.



The final part filleted .025 inches to make it easier to insert into the gear.

What did I learn?

This is my first time ever designing a part in which is for 3d printing and I had to account for tolerances in my design for the first time. This is also my first time making an extension of an already existing part all my polycarbonate sketches are just usually gussets for things like x-drive pods or to secure something at directly 35 degrees or whatever is needed on a triangle bracing system. I learned how to effectively use the project geometry tool that is offered in inventor, though I have used it before just following tutorials now I have had to use it on my own I learned a new use case for it and where to use it elsewhere. It is probably the most useful tool I have used for this project as I would have had to manually sketch the part and swap between files. I also learned of the possibility of making a part within an assembly meaning I can make any part relative to what I already have done which cut the time making this down drastically. This cad seems really simple but I mainly learned ways to make it very more efficient and that is mainly what I took away. In the real future learning how to optimize things will be very important, the quickest way to do something. In a workplace, you will get deadlines and sometimes they will be absurd and need to be met. You will have to meet those deadlines so learning the quickest ways to do so makes things easier. Another way I did this is to use the fillet tool vs manually making a 3d sketch and filleting the whole part, you have to be aware of all the tools in front of you before working to see how things can be made more efficient, such as using the project geometry tool. To put it all together I learned how to be more time-efficient while learning new ways to do the same thing. To add to this I have never 3d printed my own part before so I had to learn how to use the software efficiently and effectively. My first roadblock is that printing software used mm instead of inches. I then had to figure out why my inventor file was actually too small and not to scale at all before I printed it. I soon figured out that if I click options when exporting you have the option to change the scale units in which you can make it mm. There was more to learn about 3d printing than CAD with this experience it was really enjoyable!

The Part Printed:

The part took around 15 minutes to print, on the first print because of tolerances it was actually too big to fit inside so we had to use the thicken tool in the opposite way to cut material we ended up cutting 1mm off each side and it printed perfectly. We then made a test bench to see how much

friction it had on the screw hole and how well did it actually work? We used two 36 tooth gears with a 12 tooth on a dead shaft sandwiched in between, to our surprise there was less friction than even expected. After making it thinner it fits inside the hole almost perfectly.

