Spaceship from Rick & Morty

http://41.media.tumblr.com/3d1efe1fd1f9da89de79c6dcf6452bc/tumblr__nsq9wvllvx1uphg

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I chose to recreate this ship in Solidworks because I felt that it would be a good way to test my abilities and I felt that it was appropriate given the amount of time I had to complete it. I also chose this project because it is from one of my favorite adult cartoons, Rick & Morty. Even though the actual spaceship itself is pretty unrealistic, another way that it interested me was that it is a spaceship, and my intended major is aerospace engineering.

My process started with the base of the ship itself. For that I did a revolve. From there I moved on to make the trash can on the side. This was actually two parts. For the base a made a circle and then a series of semicircles within that to create the ruts. After cutting out the excess pieces from the outer circle, I extruded the base of the can to the desired length. Then, I did a simple extruded cut to hollow it out. For the top of the trashcan I made two circles matching the inner and outer diameters of the base of the can and extruded those. The on the side I made the handles which were just 3 different lofts. Then, I assembled the complete trashcan. After the trashcan I made the flashlight that acts as the headlights on the front of the ship. This was also an assembly of two parts. I made a circle and extruded that for the head of the flashlight then cut out a smaller circle. The base was simply an extrusion of a square. The
hardest part was the handle. That was a series of lofts to figures that I made above the base. After those lofts, I filleted the edges to make it smoother. After that I went back to the base of the ship and cut out holes for the wheels. Then I made the wheel which was just an extrusion of a circle. Then, I made to circular cuts on each side. After I assembled the wheels to the base of the ship I made the glass that is on top. For that I did a revolve with a flat line on the bottom. Then I made the base of the glass (made of metal). This was an extrusion of 2 circles. Then I cut out six holes for the screws. The screw was another revolve with an extruded cut on the top. I then assembled the screws to the base for the glass. Finally I assembled the glass with the base and screws to the ship. Then I made the yellow lines that are in the base of the ship. These are just extruded cuts. The final phase of this project was figuring out how to get the 2 flashlights and trashcans on the actual ship. I decided the cut holes into the flashlight, the trashcan, and the ship. Then I made two different poles to stick into each of the holes (I made 2 different poles because the size of the hole in the flashlight was smaller than that in the trashcan). Finally, I mated all the poles with each appropriate hole.

The hardest part of this project was creating the handles for the trashcan. I had to use multiple planes that were offset from various points. I kept having to change the distances of the planes because when I lofted it just didn’t look right. Also, I originally made the trashcan too small in comparison to the spaceship. So, I had to scale the trash can base and top. When I did this it messed up the handles and I had to recreate them. It was also annoying because I had two circles that I wanted to loft to on the same plane but I had to do 2 different lofts. After I did one loft the second circle would disappear and it was a hassle to get it to appear again.

A new feature that I learned is the circular pattern tool. I used this on 2 different occasions. I used it when I had to cut the holes into the metal base of the glass and when I made the yellow ruts on the base of the ship. This is a very useful tool and it can save a significant amount of time.

Considering that I’ve never used Solidworks before, I learned a lot from the Solidworks portion of this class. Solidworks is a very useful program and I am glad that I know how to use it now. Going into the class I had a very vague idea of what to expect because I had experience with AutoCAD, which is somewhat similar. I think having a Solidworks portion of EDSGN 100 is useful and important.