Solidworks Ferris Wheel

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I chose to create a Ferris wheel for this project because I thought it would be an interesting design to create with all of its small parts, like the carts of the Ferris wheel. Also, Ferris wheels remind me of summer fun and spending time on the boardwalk at the Jersey Shore. When crossing the bridge to get to the beach, the Ferris wheel could be seen from miles away and became a sign that told me I was close to the beach.

To actually create the Ferris wheel, I went through many different processes to create each small detail. I started with the smallest part-- the cart. To create it, I sketched only half of the floor, center pole, and ceiling of the curve and revolved it 360°. Next, I sketched half of the walls and seat of the cart, and revolved them only part of the way to leave a gap for entering the cart. To make the door, I created a new plane x inches from the gap of the wall and revolved the sketch around the center. I connected the door and the cart by making boss extrudes that interlocked together. Finally, to finish the cart, I sketched two circles and connected them to the roof, then extruded it in both directions as the piece that would connect the cart to the Ferris wheel.

Next, I designed a bar that would fit inside of the boss extrude on top of the cart and connect to the Ferris wheel once it was created. I sketched a circle 4 inches in diameter, the same diameter of the part of the cart, and extruded it to be 102 inches to make the bar. I had planned to make the two large circles of the Ferris wheel 100 inches apart, so I made the bar an extra 2 inches long so I could assemble it into grooves in the wheel that I would later create.

Following the bar, I created the wheel. I started by sketching 8 lines all intersecting in the origin, equal in 500 inches in length. I made all of the lines 22.5° apart so they would be equal distances apart. Next, I drew lines creating rectangles on each side of each line and dimensioned them to 5 inches each. I deleted the original center lines of each rectangle so they would all become 10 inch rectangles, still intersecting in the center. I then connected each of the rectangles and surrounded them by an outer circle. I also drew a circle in the center of the design where the rectangles overlapped and trimmed the lines on the inside of the circle so I could extrude the sketch. I later deleted this circle and connected the gaps of the rectangles with lines instead, and sketched another smaller circle on the inside and extruded it separately from the other parts, making it stick out farther. I then created another plane 100 inches from the front plane and repeated the process for making the first wheel. To connect the two wheels, I sketched a rectangle on the right plane equal to the radius of the circles drawn on the center of the wheels and then revolved it. Next, I drew a circle on the front of the center circular boss extrude and shelled through the entire Ferris wheel so I could create an area for another cylinder for the wheel to rotate about. Lastly, I drew circles on the inside of each wheel and made a one inch extruded cut on each circle as a place for the bars to fit into the wheel.

As the final part of the wheel, I created the cylinder for the wheel to rotate around, connecting to supports for the wheel. I began by sketching a circle on the front plane and extruding it so it was longer than the two wheels put together. Next, I created a midplane using the top and front view planes as references. This midplane was too steep, so I created another midplane using the first midplane and the front plane as a reference. I sketched the supports of the ferris wheel, then extruded them. Next, I repeated the process of creating the midplane in the other direction for supports on the other side. This plane wasn't in the correct spot for the supports, however, so I once again created another plane that intersected the boss extrude where I wanted the legs to be. I copied the sketch from the first legs and pasted it onto the new plane,
then extruded that sketch. The legs of the stand were not flat on the ground because the were extruded on an angle, so I drew a rectangle on the right plane that would even out the legs with the ground. I made this rectangle an extruded cut through all, and completed the stand of the ferris wheel.

Finally, I assembled all of the parts together. I began by mating the bars into the ferris wheel using coincident mates. Next, I mated the carts with the bars, also using coincident mates. Lastly, I mated the ferris wheel with the center axis and supports using a coincident mate so the ferris wheel could rotate around it. I created an animation by adding a motor to the ferris wheel axis to show how the ferris wheel would operate, as included in the zip file.

The hardest part of this model to complete was the wheel and the stand. I had to try many different times to figure out how to make the spokes of the wheel divided evenly, while still intersecting through the center. Every time I tried to dimension them, the lines would move from the center. Also, I originally started by making the circular parts of the wheel first and later adding the lines, but I had trouble with that, so I had to start all over. The stand was difficult because at first, I could not figure out how to make a midplane correctly. Once I figured out how to make it, I realized I had to make another midplane in order to make it at the correct angle. I also had trouble with drawing the legs of the stand because I was not used to drawing on a midplane. At first, I could not figure out how to make the legs come in contact with the center of the circle. I finally figured out that if I used the split view and divided it between the normal view of the midplane and the front view, I could see what I was doing better. Creating the wheel and the stand were the most difficult part because I had to try many different techniques before I found one that worked correctly.

In the solidworks portion of this class, I learned how detailed and complicated designs are. When making a prototype, it is a necessity to be very specific of dimensions, angles, shapes, etc. or the entire design could be thrown off. I also learned how much patience and time must go into complicated designs because they require a lot of trial and error. It is not as easy to create designs in solidworks as it may look, because it can be very difficult to figure out how to translate an idea or a shape into a computer programs.
Photos for inspiration:

- teachers.egfi-k12.org
- www.gforceamusements.com