A Newton Cradle on an Ancient Book

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I chose this object to create in SolidWorks mainly because of two reasons. One is because I wanted to create a device that I can animate simply to explore and to learn the feature, however I experienced a couple of difficulties in getting the spheres of my object to move in accordance to the right timing of a standard newton cradle. Secondly, the object relates strongly to my physics course since it exhibits the conservation of energy and momentum and I found it very interesting to design a device that demonstrates those features.

To create the sphere, I sketched a semicircle with a central axis along its diameter. After, I used the ‘revolve’ command to revolve the semicircle 360 degrees about the central axis.

To create the book, I sketched the front view of the cover. After, I used the ‘boss-extrude’ command with a depth of 12 inches to get the length of the book cover. I also used the ‘boss-extrude’ command to create a block inside of the cover, which represented the pages, with a slight offset of 0.25 inches from the edges of the cover.

To create the thread, I sketched half of it and used my new feature ‘mirror entities’ to create/get the second half of my sketch. Lastly, I drew a circle on a perpendicular plane to the first sketch with a diameter of the appropriate thickness of the tread and used the ‘sweep’ command extend this circle along the entire length of the sketch.

To create the legs, I sketched a circle and then used the ‘boss-extrude’ command to construct a cylinder to a length of 5 inches which corresponded to the length of the legs. Lastly I used the ‘cut-extrude’ command on a construct two holes in each of the legs to fit the frame of the newton cradle.

To create the long and short sides of the newton cradle I sketched a circle followed by the ‘boss-extrude’ command to create a cylinder of appropriate lengths. The ends of these parts comprised of a cylinder of a smaller diameter to fit into the holes of the legs. On the longer side, I used the ‘cut-extrude’ command to create 5 holes along its length to fit the treads.
The hardest part of this project was assembling my parts and mating them together. The dimensions relating to the treads give me trouble to mate with the frame since the dimensions had to be accurate for it to work and look appropriate. At first, my threads were too short to connect with the opposite side of the frame but with a couple adjustments and exploring the program this was rectified. Also, mating the sphere with the thread was a bit tedious since it was difficult to get the sphere to rest flat on the thread. I adjusted the thread to have a flat base to resolve this issue.

From my SolidWorks portion of the class, I learned how to sketch on different planes, how to use the different features such as boss-extrude, cut-extrude, revolve, sweep, etc., how to create an assembly of parts and how to construct a drawing consisting of the different views.
Image Portraying One of the Spheres of my Object Extended

Image Portraying the Front View of my Object
Image Portraying the Right View of my Object

Image of where I got my ideas for my project
Drawing of the Top, Front and Right View of my Project

References:

Video
https://www.youtube.com/watch?v=0xzoKf-afp4

Image
https://www.google.com/search?q=picture+of+newton%27s+cradle&espv=2&biw=1920&bih=935&source=lnms&tbm=isch&sa=X&ved=0ahUKEwiDl rXWo6PMAhVLFj4KHTsNBX0Q_AUIBygC#imgrc=yzEgUeGAy0DAoM%3A