STEALTH BOMBER
SOLIDWORKS PROJECT

~Brad Hensler ~ EDSGN 100 ~ Section 22 ~
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About my project:

When I first heard about this project I was not really sure what I wanted to create using the SolidWorks program. Originally, I thought that I either wanted to do a football stadium or Iron Man helmet, but eventually I came up with my final idea to create a stealth bomber model. I chose this object because I’ve always been fascinated with how they work and what they can actually do. On top of that, I also have a lot of family members and friends that are serving in the military so I’ve always been up to date with what is going on with the wars and am very interested with everything it involves.
Furthermore, the completion of this stealth bomber involved many different features to get the final object to look as much like the real thing as possible. First, I started with the base by extruding up the general shape of the plane. I then filleted the back edges of the plane and made sure the wings were at the right angles. After that, I used the loft feature to create the nose part of the plane which was pretty challenging to actually get it to look the right angle. I also lofted the back end, but had to also shell it to make sure it was cut at the right angle, which is just one of the new features that I never used before. When that was completed, I started working on the side trapezoid areas which were definitely the hardest part of my project. During this step, I had to somehow produce that 3-D trapezoidal shape that had its face coincident to the face that it was next to, as well as slope back down towards the back of the plane. This required a new feature, called the draft feature which I used along with a highly modified loft to produce the exact shape that I wanted. When I produced one of those trapezoidal shapes, I then mirrored another it to the opposite side, which was another new feature that I used to duplicate the exact shape instead of trying to reproduce it again. Next, I moved on to producing the tail, which I completed by creating two more parts and lofting identical rectangles on an angle to produce it and then assembled the three parts together in the end. Lastly, I added a black iron material to the whole assembly, then used a new feature in the appearance tab to add the decals.

In the end, the SolidWorks portion of the class has taught me another great way I can perform modeling for design processes, as well as other real-life applications. It is an easy to use software that can bring my ideas to life, and I know in my future, not just here at Penn State, but once I go on to pursue my career that I will be using this software to help prototype and better represent any drawings, brainstormed ideas, etc. that I may come across.