

Project overview essay examples

[Transportation](#), [Airlines](#)



INTRODUCTION

This is the report of the ME202 Summer course project performed in 2012. The first part of the report is the project overview followed by the sketching activities, CAD modeling activities, and then Problems & Solutions. The last part of the report is the Conclusion which was derived after the project has been completed. All the engineering parts, assemblies, and drawings are shown in the appendix.

This project involves the manufacture of a bicycle using the basic knowledge and skills derived from Solidworks. For this task to be complete, different parts of a bicycle must have been modeled and assembled in Solidworks. The parts include the seat, the rear gear set, the front gear, the tyre, wheel, front fork, and the bicycle frame. An AVI file of the exploded assembly is also produced.

This project is made to ANSI standards. All parts are made with a dimension of 5'6" and on a 1: 1 scale. The dimensions are in millimeters, the in seam size is kept at 781. 05mm, the frame at 530mm and the length of the top tube at 540mm.

All the engineering drawings shown at the appendix were made in accordance with the First Hand Projection System. The appendix also contains two isometric prints of the 3-dimensional representations of each component, one of these prints has all dimensions indicated on it while the other hard copy print has no dimension indicated. It also contains one isometric print of the completed bicycle in 3-dimensional view.

SKETCHING PREPARATIONS

Before any feature can be created in Solidworks a sketch is needed first. All base features are essentially sketched features which are created by using a sketch. A sketch can be simply defined as a combination of entities such as lines and arcs. Sketches are made in sketcher environment by:

- Invoking the sketch command and using it to build features.
- Entering the sketcher environment by default via the invocation of the feature-based command.

The Solidworks' sketching environment can be invoked at any time during the course of part assembly and when in drawing modes.

Solidworks contains several sketching tools and operations, majority of which were used in creating this project. These operations were used even when marking a point, constructing a slot, when drawing a centerline, and when constructing a circle. Relations can also be added to minimize the workdone for dimensioning. Repeated drawing of the same sketches can be avoided using commands like " mirror', ' circular pattern', and ' linear pattern.

CAD MODELINGS

PART FILES

The building block of the Solidworks mechanical design software is the 3D aspect. Feature-based approach was employed when modeling parts.

Features can be described as complex combinations of objects and operators which are considered together as a unit and can be modified or duplicated.

The " feature manager design tree" was used to keep order of operations; parametric changes can propagate through this feature tree.

The first solid feature created when a mode is created in the part mode is

called the Base Feature. The selection of the base feature is very crucial as every other feature is built based of these base features. Base features are usually created using reference planes or datum planes.

ASSEMBLY FILES

Assembly is described as a combination of two or more components using parametric relationship. These relationships are called mates, in solidworks. These mates make it possible to constrain component's degree of freedom at their respective work positions. Components may be part or sub-assembly and are linked to the assembly file. Due to the full associativity of solidworks, changes made on any part reflect in all the assemblies in which the part is used.

Assemblies are made in two ways:

- Bottom-up design- This is a traditional method. In this method, parts are created, inserted into assembly, and constrained as required by one's design. It is preferably used when a previously constructed, off the shelf parts are used.
- Top-down design- In this design method, work is started in the assembly and parts are built to fit into the assembly. The geometry of one part is used to define the other parts or to create mechanized features which are added only after the parts are assembled.

This project was made using the bottom-up design approach.

The figures below explains how the assembly of the bicycle was done

DRAWING FILES

An engineering drawing, a kind of technical drawing, is used to fully and clearly define the requirements for engineered objects. It is not just the drawing of pictures; rather it is a language, a graphical language that communicates information and ideas from one mind to another. A set of product related engineering drawings and corresponding lists related to design, inspection, manufacture and test of an item or system is called an Engineering Drawing Package(EDP). This EDP should contain the required details to authorize design evaluations and cost protuberances at the initial stages of a project.

As the design and its development advances and the project moves through the engineering and manufacturing development phases into the production the drawings would consequently get more complete. The EDP in concurrence with the System, subsystem, or the equipment specifications should contain sufficient information that would allow any competent manufacturer to get components and materials to carry out the manufacture, test and inspect items indistinguishable and interchangeable with those that are delivered by any other manufacturer using the same EDP. The EDP encloses all the suggestive documents desired to ensure the competitive procurement of a system or object.

PROBLEMS AND SOLUTIONS

The basic problem encountered in the initial stage if the project is that is was discovered that after some features were made the base dimensions where undesirable but could not be altered at that point in time. This is due to the indication of an error in the child features whenever a change is made to the parent feature. This problem could have been avoided if the base features

had been carefully created as all the child features are dependent on the base features.

Another problem encountered over and over again is that of the “ zero thickness” error which sometimes leads to the feature failing to complete. This problem usually occur whenever there is an attempt to extrude on the surface of the base feature. It can be removed by ensuring the sketches are made just inside the present base feature.

TECHNICAL SKILLS

Working with SolidWorks teaches a blend of precious mechanical CAD, design justification, and data management skills as one construct parts, assemblies, drawings, and so on. The skills acquired include: measuring, sketching, documentation, common commands such as the sweep command, mirroring techniques, using geometric relations, dimensions, planes, and several assembly commands. Also, this project requires one to present the project in an organized format.

CONCLUSION

WHAT I LEARNT

I learnt various mechanical, technical and design skills during the course of this project. I learnt how to use complex commands like loft, sweep, mirror, revolve and many more thoroughly. I learnt the basic practical knowledge of making parts and assemblies. I also learnt the basic skills of designing. The EDP aspect taught me how to make detailed sketches of parts to be manufactured.

The experience I had while battling with the above mentioned problems such

as the zero thickness error and the dependence of child features will come in handy in future projects. This is because as at the time the project gets completed I would have been able to fully comprehend and consequently have a full command over the software thereby not encountering these same problems again.