ECE 480: Design Team #9
Application Note
Designing Box with AutoCAD

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Executive Summary

This application note demonstrates how to design a box to hold a video recording tool in NX 8.5. NX 8.5 is a computer-aided design (CAD) software used to create technical drawings. The video recording tool has a lot of wiring that goes with the cameras, display, and microcontroller. In order to make the whole project presentable, a box is designed in AutoCAD to hold the display and all the other components.

Keywords

Computer-aided design, NX 8.5

Introduction

Computer-aided design (CAD) softwares are used to create technical drawings for many engineering projects. They allow a user to design faces, curves, surfaces, solids, and much more. A 3D printer can be used to bring to life what was designed on the CAD software.

CAD can be used to design something as simple as a surface, but also as complex as a perfume bottle. For this project, we needed to design a box of some sort that could house all the components of our project, while also having the ability to mount the display. Then, based off the cost of the design, we as a group will decide if 3D printing is feasible or if we should construct our own version of the ‘box’.

There are many different CAD programs. The program used for this project is a software by Siemens, called NX 8.5. This software is widely used in the industry to create 3D models of various types of objects.

This application note will show how to use the basic features of NX 8.5 such as creating faces, solids, and constructing holes.

Procedure

Create New Design Work Area

First step is to open NX, which can be done with the following steps:

1. START menu
2. All programs
3. Siemens NX 8.5
4. NX 8.5

Then, by selecting NEW in the top left corner of the main screen, the image shown in Figure 1 below should pop up. Select the correct units (inches) and the correct folder to save your project, and click OK. This will open up a new design work area, shown in Figure 2.
After you reach the image in Figure 2, you are now ready to start designing! The next step is to design a face and then turn it into a solid face. First, one must decide on the shape of the face. For this project, we needed a rectangle.

**Design Rectangular Solid Face**

There are three main steps to creating a rectangular face: sketch, rectangle, and extrude. Click on the sketch button in the bottom of the screen, (shown in Figure 3 below), and the ‘Create Sketch’ box will appear (shown in Figure 4).
The most important part of this process is selecting the plane you want to create your face in. You are given an XYZ coordinate system to choose from (shown in Figure 5).

In the Select Planar Face or Plane option, you must hold your mouse on the plane you would like to choose, and click on it. If done correctly, a ‘1’ should appear in parentheses like so:
Then, select the rectangle shape shown on the bottom of the screen (Figure 7).

Based off the dimensions you want your rectangle to be, draw the rectangle. An example is shown in Figure 8.

You can change the lengths and widths (5.8 x 3.171 inches) any time just by clicking on the values. After completing this step, you must click on ‘Finish Sketch’ which is shown on the right side of the ‘Sketch’ option you chose earlier (Figure 9).

The next step is also extremely important, extrude. This step will turn your face into a solid (with depth). Click on extrude, which is shown in the top left corner of the main screen (Figure 10). This will pull up the screen shown in Figure 11.
You must select the thickness of the face you would like. For this project, 0.25 inches was selected. You can input the thickness value in the ‘distance’ blank. Further, you must select the direction you want the thickness to be in the XYZ coordinate system. After doing so, click OK. Now, you should have completed the three steps to construct a rectangular solid face on NX 8.5! An example is shown below. You may rotate and pan your image in the ‘View’ menu for a better look on your rectangular solid face.

By repeating the three steps described above, we created a few rectangular faces, designed next to each other (shown in Figure 13).
Construct Hole in Face
The next step was to how to create holes in NX 8.5. You must select the ‘Hole’ option in the top of the screen.

After choosing the diameter of the hole, the location, and other parameters, you can click extrude and select the depth of the hole. For the purposes of this project, we needed a complete hole in the face. So the depth of the hole chosen was the same as the depth of the rectangular face created earlier.

An example of what a face should look like with holes in it is shown in Figure 15.
Similarly, you could also create holes in the shape of rectangles. It can easily be done just by the three step method described above of creating rectangles, however, just design the rectangle in the location of the hole, and extrude. Two angles of the design with holes in the shape of circles and rectangles are shown in Figure 16.

![Figure 16](image_url)

**Extra Features**

Described above are some basic functions of NX 8.5. These features will be available on most other CAD softwares. There are also many other features available, such as shelling a solid to make it hollow. Shelling is basically another way of creating a hollow box. It is actually a lot simpler than creating a box face by face (described in this report). However, with the design we needed, going face by face was a lot easier, especially since every face was different.

Another feature available in CAD programs is edge blend. This feature allows a user to blend the corners of edges of a face to make it look more presentable. The radius of the blend as well as many other parameters can be selected to make your design exactly the way you need it to look.

**Conclusion**

After some minor tweaking, the design will be exported as an .stl file and given to the Division of Engineering Computing Services (DECS) to 3D print. Before actually printing, the group will obtain a rough estimate on how much it will cost to fabricate this design. Based off the estimate, the group will decide if they want to fabricate the box using a 3D printer, or go about another method of making the box.

Regardless of 3D printing, using AutoCAD is an extremely useful skill to have in the industry. This application note explains the basics on how to use Siemens NX 8.5. There are various other AutoCAD softwares, however they all have the same basic functions that were described above for NX 8.5.
References

Siemens NX 8.5

http://www.autodesk.com/solutions/cad-software

DECS at MSU