

# Enclosure Prototyping Using Siemens NX 8.5

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## Abstract

When prototyping electronics, it is often necessary to develop an enclosure to house the components of the project. This holds the parts together, as well as protects them. Engineers utilize computer aided design (CAD) software to create digital representations of the enclosures. Having a visual design helps the engineering team to understand the characteristics of the final product. In addition, the CAD file provides instruction for a machine to fabricate the case based off of the engineers design.

One of the most popular CAD software applications available is Siemens NX 8.5. The program is widely used in industry by professionals and is a comprehensive tool for creating 3D computer models. This application note provides step-by-step instructions on how to use NX software in the enclosure design process. It teaches the reader specific software commands to use and also gives tips that will allow the engineer to leverage the power of NX to build an enclosure that most meets their needs. Major topics covered include visualizing the final product, getting started with NX 8.5, design techniques, advanced strategies, and the final product. Covering these topics provides the user with enough knowledge to immediately begin designing their own enclosure for projects.

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## Introduction

Proper housing for electronic components is an important part of product development. It can be especially challenging for an electrical/computer engineer who may not be familiar with CAD software. Even so, before immediately learning how to use a CAD program, it is important for the user to understand the purpose of the software which will be used.

NX 8.5 is a modern iteration of Siemens professional software package. Direct applications for the product are to assist engineers in design, engineering analysis, and manufacturing. Licensing of the product is proprietary, although engineering students at Michigan State University have access to it in computer labs.

This instructional document is intended for a person who is completely unfamiliar with engineering an enclosure using NX 8.5. University students may especially find this instructional useful, as they may not have had exposure to CAD software in the past. Engineers of all disciplines can take advantage of this skill to further their professional development and enhance their ability to communicate with designers that use this software. By the end of the tutorial, the user will be acquainted with the basics of NX 8.5 software and will learn useful techniques for successfully developing a robust enclosure design.

## Section 1 - Visualizing the Final Product

Before a CAD drawing is made, the final product should be visualized by the engineer. It is helpful to draw a rough sketch of what type of enclosure you would like to create. NX is very flexible and can produce any 3D object. Additionally, measure the dimensions of the components so that you know their dimensions. It is extremely important that the design engineer understand the physical characteristics of the enclosure internals, so that a correctly scaled case can be created. Lastly, compile various potential designs and write down the pros and cons of each.

## Section 2- Getting Started with NX 8.5

Engineering students at Michigan State University can access NX 8.5 by visiting the programs within their computer workstation. Go to *All Programs > Siemens NX 8.5 > NX 8.5*. Figure 1 illustrates the location.

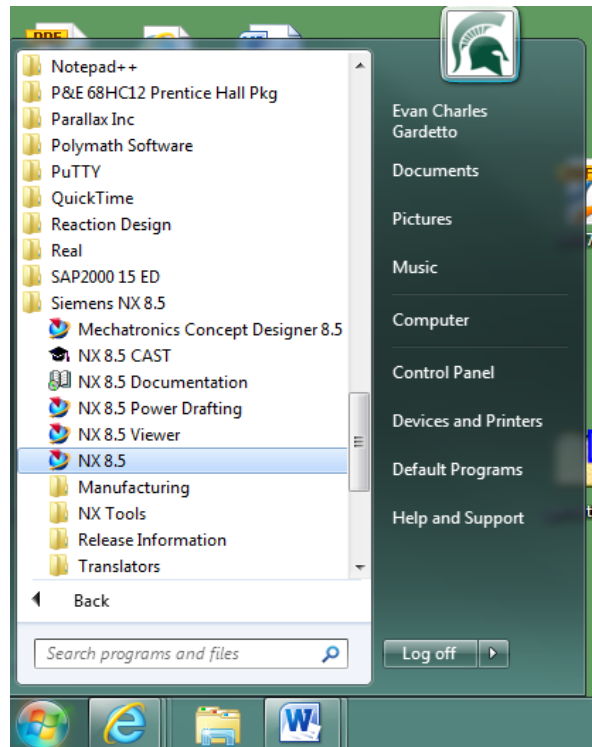


Figure 1

After selecting the icon, the program will boot up. The opening screen will appear, giving access to the many available commands in NX. Figure 2 shows what the user will see when the program opens.

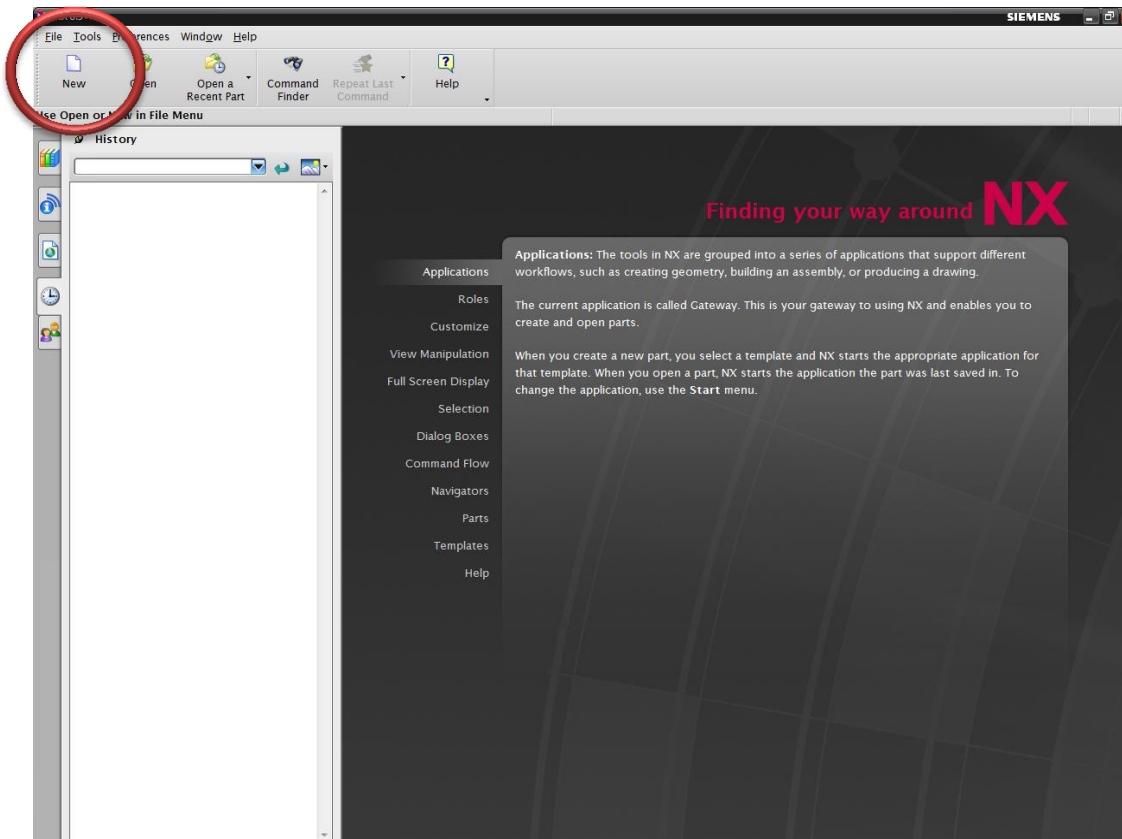
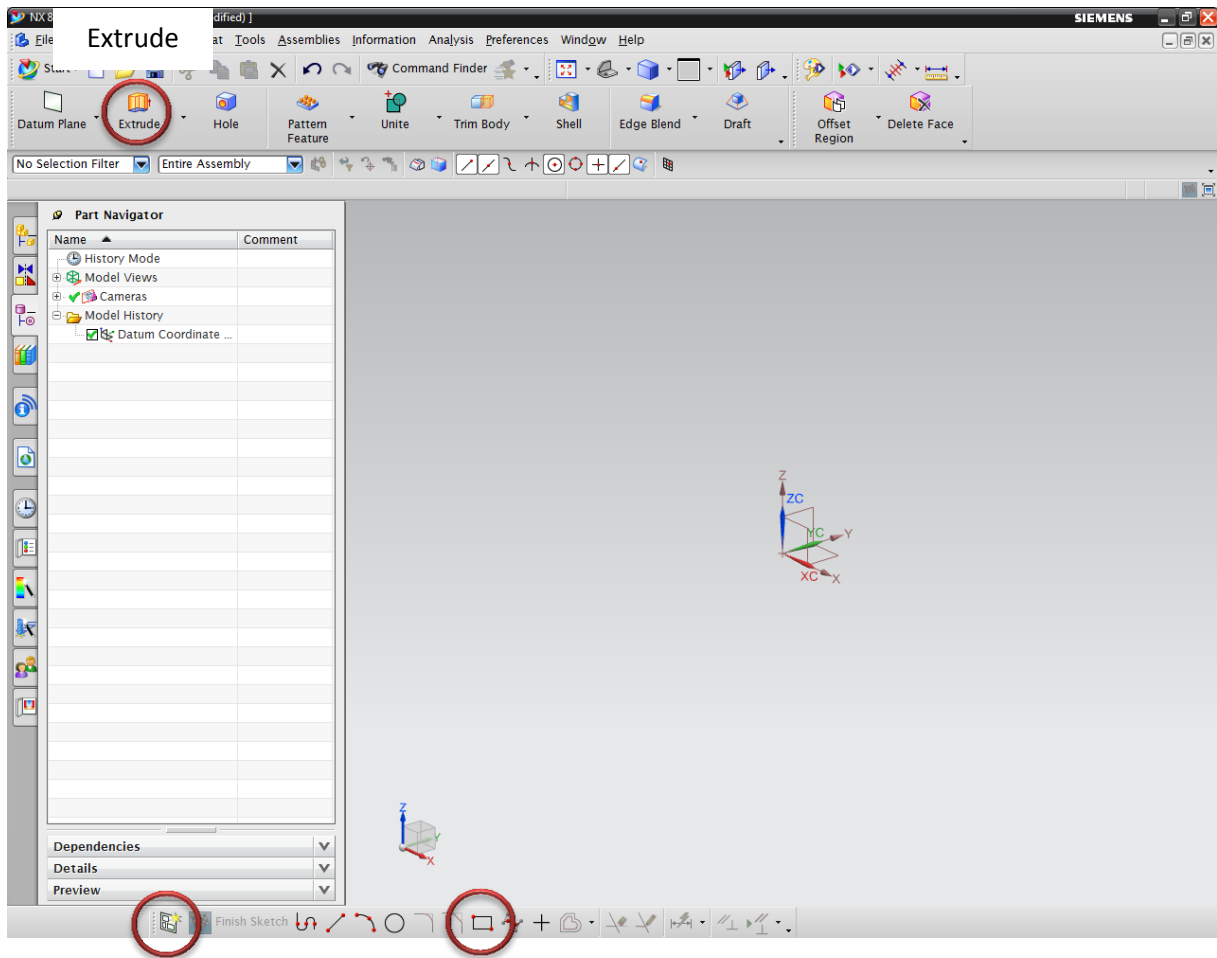


Figure 2

Look around the main screen and become familiar with the available buttons. Click on *New* to begin a sketch. Select *Model* under the *name* column. Give the file a name and select OK. NX will create a new design work area, and we are ready to begin.



Sketch

Rectangle

Figure 3

Fundamental design concepts will now be covered to teach the key functionalities of NX 8.5. As displayed in Figure 3, the coordinates of the work area are represented in three dimensions using the x, y, and z, coordinates. This gives the designer constant orientation awareness. When making a drawing, sections are produced piece by piece, and then extruded together to form the final product.

The first step in learning how to use NX is by forming a block that can be manipulated on the screen. Select *sketch* (Figure 3) in the bottom left portion of the screen. This initiates the drawing process and allows various shapes to be implemented into the design. In order for NX to know which direction the element should be oriented on, one of the three planes (x, y, z) must be selected. Choose a plane and select the *rectangle* (Figure 3) option in the bottom toolbar. Use the cursor to draw a square and then click *extrude* (Figure 3). A 3D block will appear in the work area similar to Figure 4; this is one element to the design.

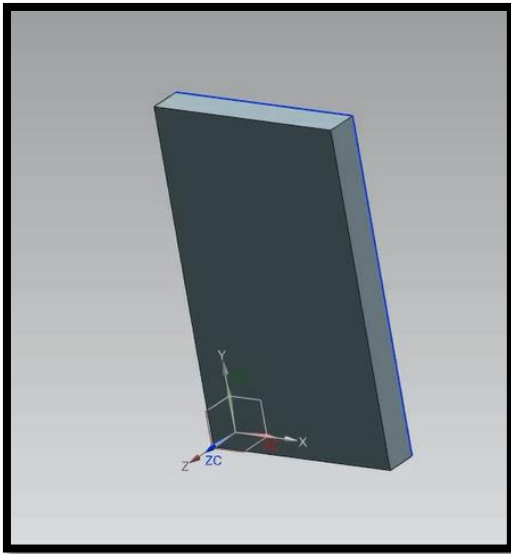


Figure 4

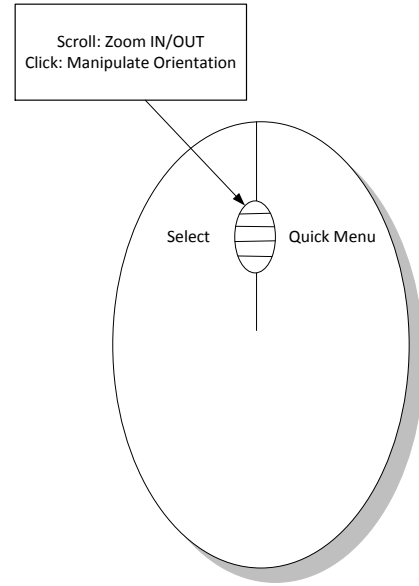


Figure 5

Manipulation of the element is done using the mouse buttons. Scrolling with the mouse will zoom in and out on the cursor. Clicking the scroll button will allow the user to pane around the object to view from 360 degrees. Viewing the object at various angles is critical to ensuring that everything has been designed properly.

Next, another block will be added to the design and extruded with the first element. In order to do this, hover the cursor over the block and select a plane to connect the new segment. Click on the *sketch* selection once again and use *rectangle* to draw another block. After the *extrude* button is pressed, the work area will show the new block connected to the first one. Figure 6 shows an expected output.

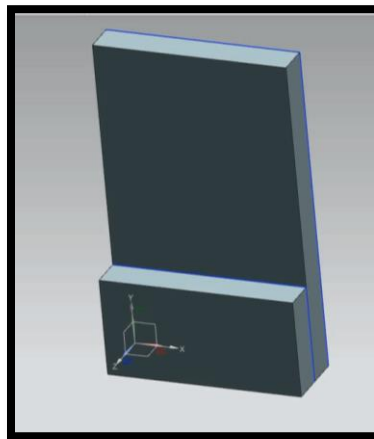


Figure 6



With this knowledge, the user is now capable of implementing basic concepts regarding enclosure design through NX. Sketching more blocks into the work area can produce a primitive housing. While this establishes fundamental concepts, the instruction will now focus on design techniques used to create an optimal design.

### Section 3 - Design Techniques

#### Shapes

The NX interface provides users with a toolbar of design tools at the lower left portion of the program. The rectangle selection was chosen earlier, but there are many others options to use. Figure 7 shows many of the choses that the user can utilize. Some of the most useful are line, arc, circle, and rectangle. With these, the designer can easily draw whatever shape they can imagine and manipulate it in the work area. Experiment with various choses using the same steps in the rectangle method used earlier to see what shapes can be created.



Figure 7

#### Dimensions

The design of an enclosure require significant precision. NX allows the user to input precise measurements into the elements. To do this, simply double click a side of an element and the length/width input box will appear (see Figure 8). Specify the desired measurement parameters and make sure that they are consistent with measurements taken of the actual components that will be housed inside the enclosure.

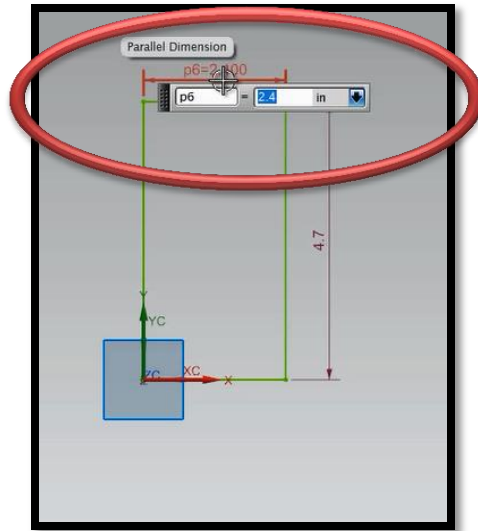


Figure 8

## Creating Holes

Often, an enclosure will require holes in the body. This could be for wire routing, external connections, etc. NX has shortcut buttons to accomplish this task. Start by clicking on the *hole* shortcut button (Figure 9) on the toolbar at the top. Now, the user can use the cursor to mark the center of the desired circle on a design segment. Multiple holes can be created at once by placing multiple marks. When all locations have been specified, click the *checker flag* to finish and something similar to Figure 10 will appear.

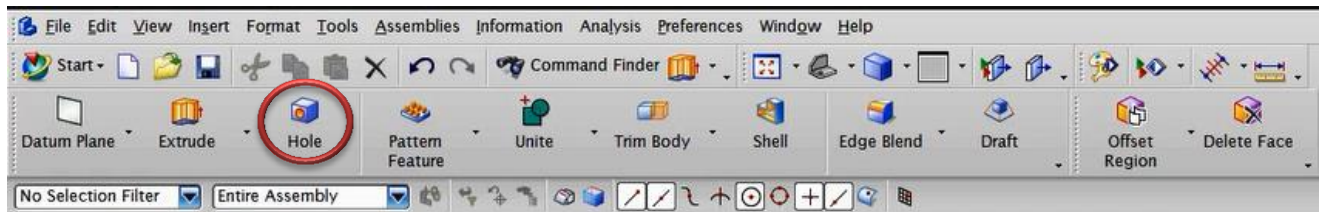


Figure 9

From here, the *hole* menu on the left side of the design window allows for various parameters to be set, such as radius, hole direction, and depth depending on the enclosures particular design specifications. When the proper setup has been inputted, press the *extrude* shortcut and the hole will be punched out, as illustrated in the transition from Figure 10 to 11. The hole diameter can be altered by clicking on the hole, then switching the parameter values in the pop up window.

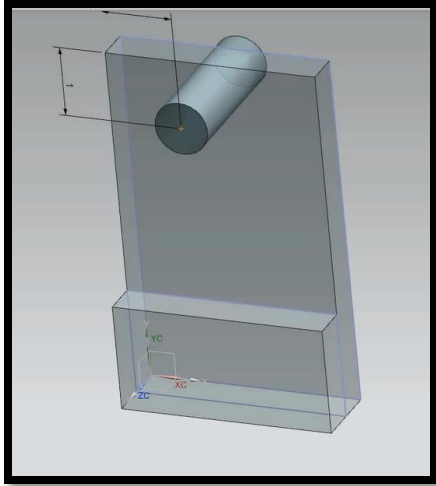


Figure 10

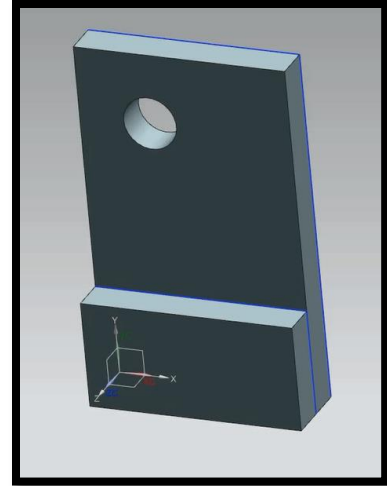


Figure 11

## Shelling

An enclosure needs to be shelled out so that there is room in the interior for components. NX incorporates a built in shortcut for shelling the design. Click on *shell* (Figure 12), and then simply hover over the part that needs to be shelled out. The face of the block that is set to be changed will highlight in red for confirmation. Click on this highlighted area and the block will become carved out. The thickness of the wall can be altered using the command menu on the left side of the screen. Continue to select walls that need to be shelled.



Figure 12

Figure 13 shows a design before it has been carved out. Figure 14 illustrates after using the shelling tool. This tool is very useful in making a custom housing. It is important for the engineer to consider the width of the wall to ensure that it provides adequate protection. Too thin of a wall and structural integrity will be compromised, too thick and material overhead and weight may become an issue.



Figure 13

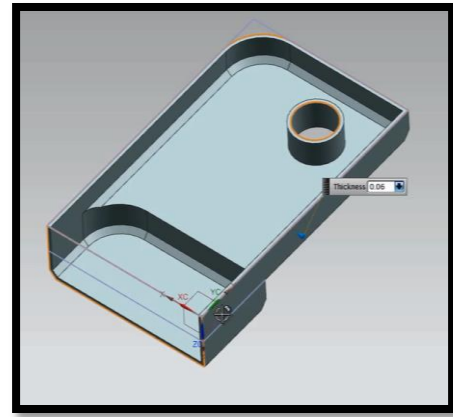
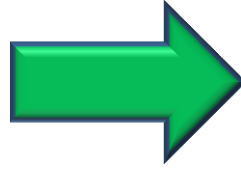


Figure 14

## Section 4 - Advanced Strategies

### Curvature Design

The blocks that are produced in NX have sharp edges by default, which may be undesirable for the final product. Edges could cut the user and can also be aesthetically displeasing. NX 8.5 offers tools to add curvature to the digital design. Two of the most useful for developing a prototype enclosure are the *Edge Blend* and *Chamfer* commands. *Edge blend* creates a smooth curve transition from one plane to another. *Chamfer* produces a flat line connecting two planes. In other words, it cuts the sharp edge off and leaves a 45 degree edge. As shown in Figure 15, the shortcuts for both of these tools are found in the main toolbar.

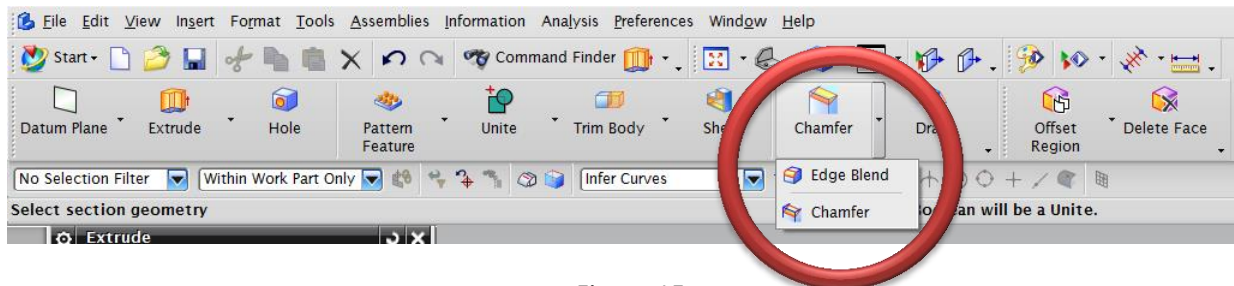


Figure 15

Figure 16 gives an example of edge blending a couple of corners on a design. It creates a nice effect and has practical benefits as well for the end user. The circular radius of the edge blend can be altered using the command menu on the left side of the screen. A smaller radius will produce a tighter curvature, while a larger radius will make the transition more drawn out.

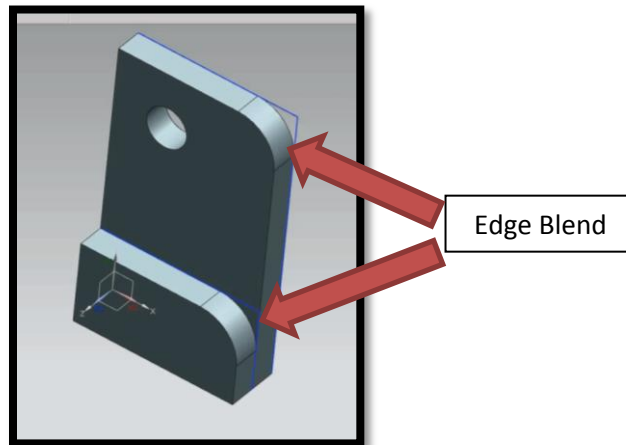


Figure 16

## Section 5 - Final Product

With the skills learned in the design techniques and advanced strategies sections, the designer has the background to produce an enclosure in NX 8.5. After the designer is satisfied with their enclosure, they can produce a physical prototype. Figure 17 gives an example final design, which has room for components to be placed in the compartment area on the right and an access hole on the bottom left side. NX 8.5 makes it simple to produce accurate digital models like this quickly and efficiently.

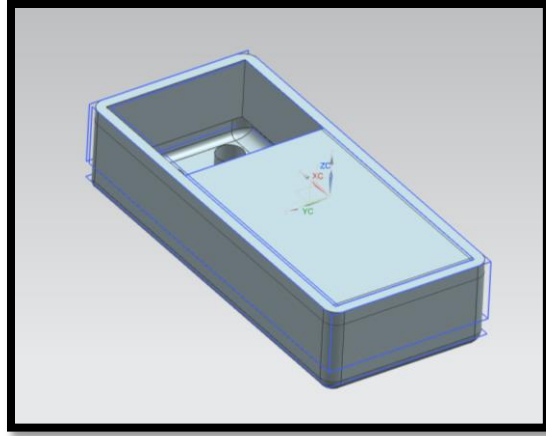


Figure 17

Many university students are taking advantage of on-site 3D printers to quickly produce prototypes from their CAD designs. For a small fee, a design team can produce plastic prototypes for projects. Once an enclosure has been printed, it can be analyzed and any necessary adjustments can be made in NX. Then, after revisions are made, the next iteration of the design can be produced until optimal design has been reached.

## Conclusion

Siemens NX 8.5 software provides engineers with the ability to create a digital version of an enclosure for their projects. This application note provided a glimpse into the potential of NX, but there is still very much to learn. Understanding how to use NX 8.5 expands an engineer's interdisciplinary abilities, promotes ingenuity in the design process, quickens time to market and overall improves the product development cycle.

## References

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[http://en.wikipedia.org/wiki/NX\\_\(Unigraphics\)](http://en.wikipedia.org/wiki/NX_(Unigraphics))