

MICHIGAN STATE UNIVERSITY

# Modeling in Siemens NX

ECE 480 GROUP 11

He Chen  
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**Abstract:**

Products which are developed with the use of Computer-Aided Design (CAD) software are ubiquitous in the current market. This paper will demonstrate the usage of NX, a CAD software developed by Siemens AG. NX is considered the industry standard for various fields.

**Keywords:** Siemens UGS NX UG 3D Modeling Assembly CAD

**Contents**

Introduction: ..... 2

3D Model Breakdown: ..... 2

    Step one: Modeling each part..... 3

        Constraints:..... 4

    Step Two: Assembly ..... 7

Resources: ..... 8

## Introduction:

Siemens NX is a Product Lifecycle Management (PLM) software that is specifically designed to aid in the all aspect of designing a product. Siemens NX can be broken down to three main uses. Computer-Aided Design (CAD), Computer-Aided Engineering (CAE), and Computer-Aided Manufacturing.

CAD involves creating a digital model of your idea based on the specific requirements of each application. CAD is the main focus of this paper. CAE involves validation of a design, this includes various simulations such as Finite Element Analysis (FEA) and Computational Fluid Dynamics (CFD). These methods can be used to validate structural integrity and/or longevity of a product. CAM involves manufacturing of a design. Once a design is ready to be manufactured CAM can be used as an aid to program various manufacturing machines such as CNC mills and Lathes to create the physical product.

## 3D Model Breakdown:

In this paper we will use Figure 1 as the desired object to model.



Figure 1

The solder sucker in Figure 1 can be modeled as a single part in NX. However this is not the proper way of modeling. A part is defined as an object that cannot be disassembled any farther. Thus a NX parts file should contain a single part. Multiple parts files can be assembled in a assemble file.

In order to model the each parts correctly disassembly of the desire part is necessary. Figure 2 shows the 14 parts that makeup this solder sucker.



Figure 2

### Step one: Modeling each part

When modeling a part, it is necessary to break down the part into simple geometric figures and then simple 2D shapes. Take the tube in the center of Figure 2 for example. Figure 3 is a close up of the modeled part.

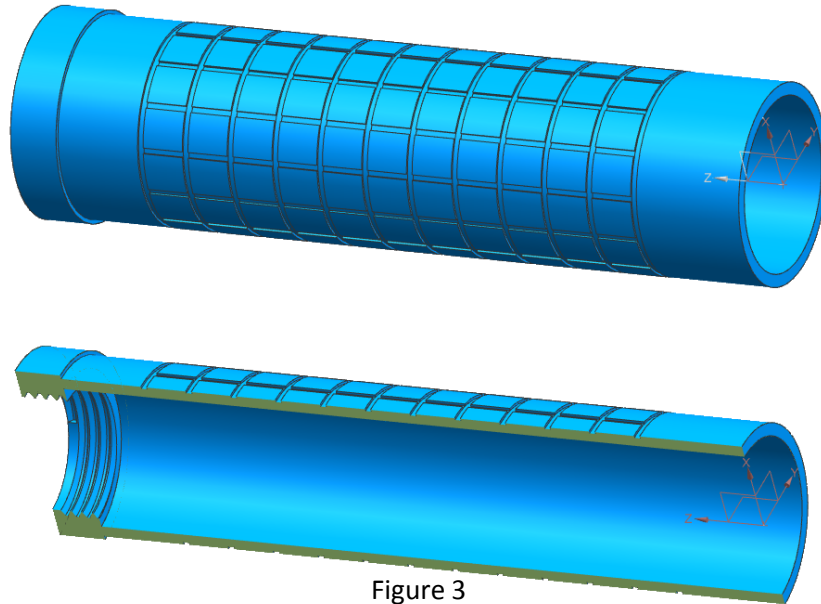


Figure 3

As Figure 3 shows, the tube housing has various inner and outer diameters. For now let's ignore the square patterns. Two can model this part in multiple ways, however extruding a 2D figure to create this 3D figure will be one of the simplest.

To create a new part in NX, we need to click File -> New -> Modeling Tab -> Model -> choose name and directory -> Click OK. When creating a 2D figure in NX, it's called a Sketch. A sketch has to be drawn on a single plane. The sketch icon can be found in Figure 4 in the red outline.

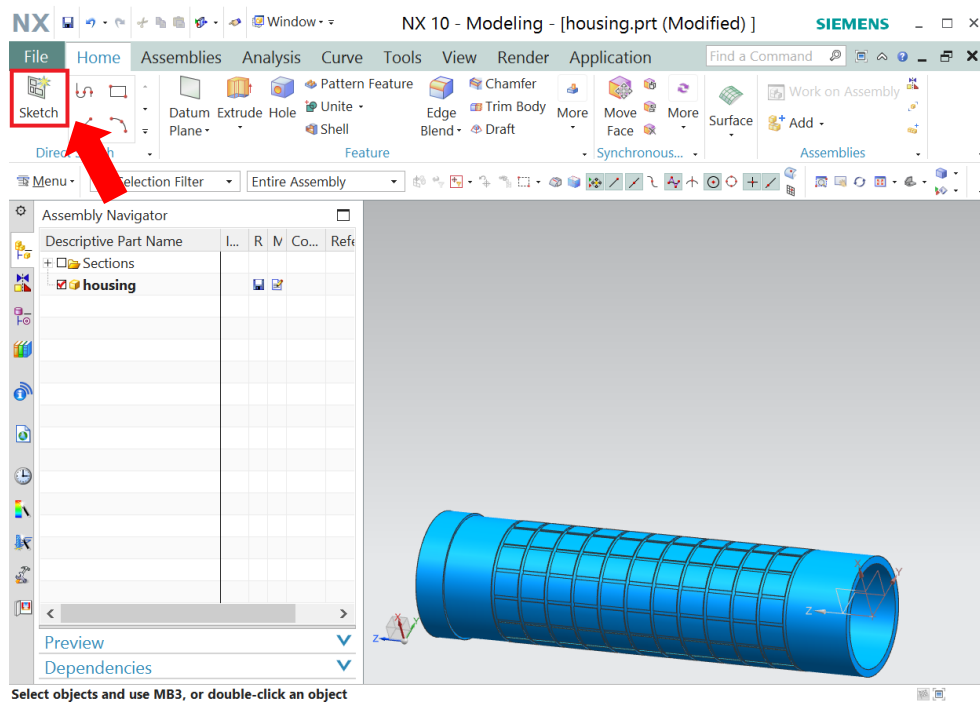


Figure 4

Once sketch is clicked a prompt of which plane the desired 2D drawing will be asked. Any plane could be chosen, however for this example we will use the X-Y plane given. Once a plane is chosen the camera will change to be orthogonal to the plane for easier drawing angle.

Figure 5 shows the sketch for a portion of the model.

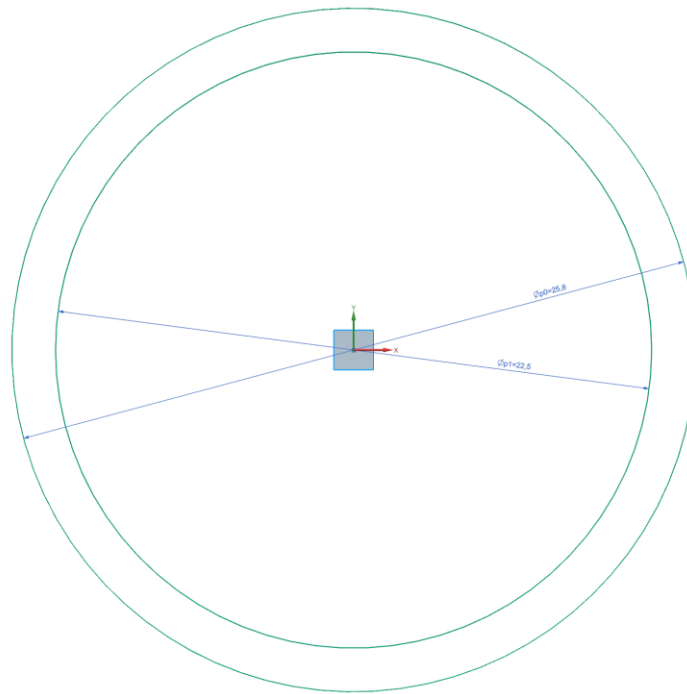


Figure 5

### Constraints:

Once the 2D drawing is completed, constraining the lines and curve is essential. NX wants a single solution for each line and curve. Thus a line need four rules, for example if we tell NX the coordinates of the two end points of a line, then that line is fully constrained where it can only be that solution. NX has two forms of constraints.

### Dimensional Constraints:

Dimensional constraints are constraints (rules) that has numerical values. Such as the coordinates of a point, the length of a curve, the radius of an arc, the angle of a line, etc. These constraints can be accessed by the hotkey "D" on the keyboard when you are inside the desired sketch. Figure D shows the window for Dimensional Constraints.

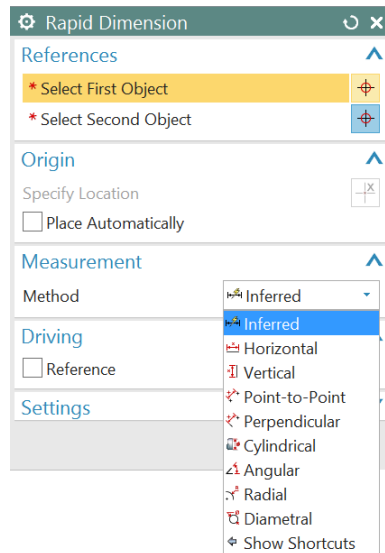


Figure D

In Figure D's window, the first two lines are where you would select one or two objects you are constraining. For example, the length of a line only requires the line itself. But the angle of a line requires the line and another line/vector you are referring to. For methods you are selecting which type of measurements you desire.

#### Geometric Constraints:

Geometric constraints are constraints (rules) that do not have numerical values. Such as midpoint of a line, having two end points touch, parallel lines, perpendicular lines, having two arcs in concentric, etc. These constraints can be accessed by the hotkey "C" on the keyboard when you are inside the desired sketch. Figure C shows the window for Geometric Constraints.

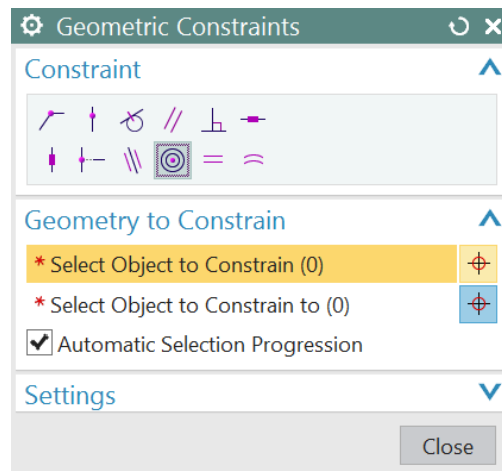


Figure C

In Figure C's window, the first selection is what type of geometric constraint is desired, they are as follows. Two endpoint touching, endpoint on curve, parallel line, right angle curves, horizontal, vertical, endpoint on midpoint, two collinear lines, two concentric arcs, equal lengths, and equal radius. The second selection are your choice of two or more objects to assign rules to.

In NX there are multiple of ways to use a 2D sketch to create a 3D figure. One of which is Extrude. Extrude will take a sketch and "pull it" on a vector. Extrude can be found in Figure 4, four icons to the right of Sketch and it's has a Orange icon. For example if we Extruded Figure 5 on the Z-axis we will get the following solid in Figure 6.

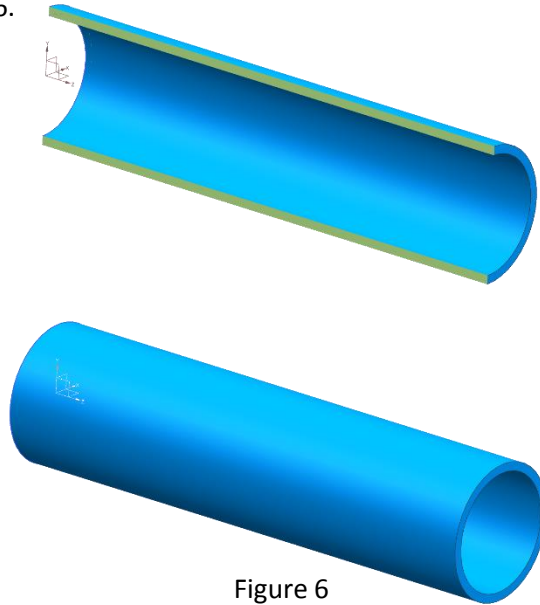


Figure 6

The next step would be to model the larger OD and smaller ID portion of the part. This portion can be done using the previous method with the end of our current tube as the plane. The result of the extrusion can be found in Figure 7.

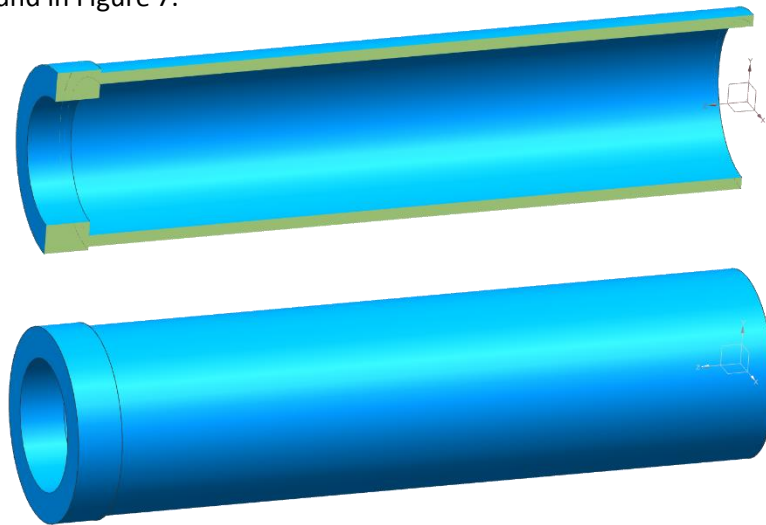


Figure 7

Figure 7 is a complete functional model of our desired part. However we can farther add detail to the object using Thread feature and pattern feature for the part. The finished product can be seen in Figure 3.

These steps can be repeated to and applied to all 14 parts of the solder sucker.

## Step Two: Assembly

Once each individual part is modeled, we can then create an assembly file. An assembly file will allow you to import in all the modeled parts and assembly them virtually. Figure 8 shows the 14 parts to the solder sucker in their proper exploded view.

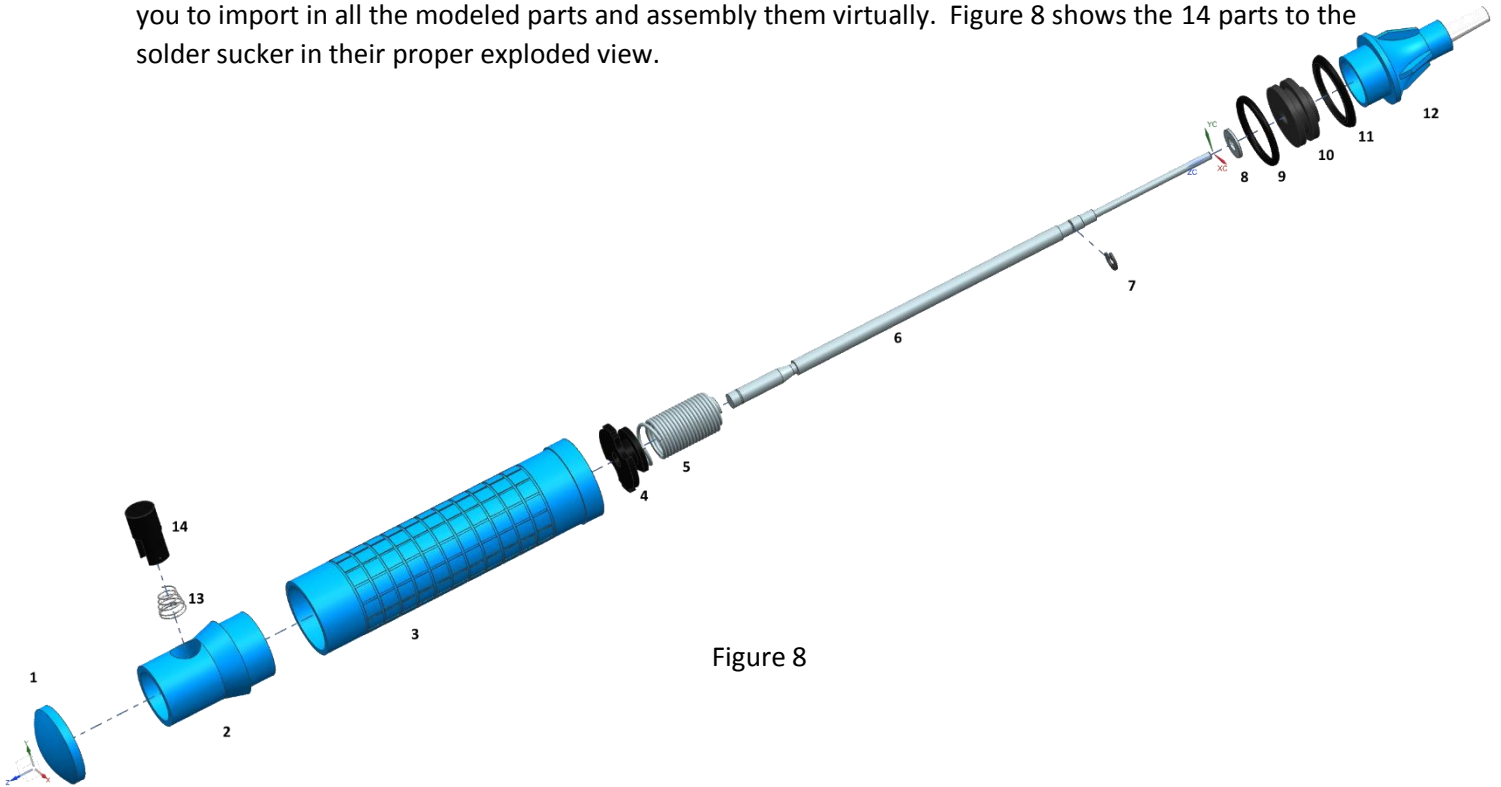


Figure 8

Once the parts are imported, you can move each part by using the “Move” option. Just right click on the part and click Move. Move the parts around to the order you wish to assembly the parts. All the parts can be constrained to other parts using Assembly Constraint rules. These rules consist of Align, Center, Bond, Fix, Angle, Distance, and many more. These rules give you a complete toolkit to fully assembly every part of this solder sucker.

For example we can “Fix” part 6 in Figure 8. Fix will bound the part to its current location, thus it cannot be moved in anyway. Once part 6 is fixed, we can constraint part 1, 2, 3, 4, 5, 8, 9, 10, 11, and 12 by aligning its center line with part 6’s centerline. This will limit the 10 parts to only have motion in a single vector. We can farther constraint the part using “Touch Align”. We can tell NX that part 1 touches part 6 at a certain point this part 1 is fully constraint. We can follow this methodology to the best of the parts and we will get the following end result in Figure 9.

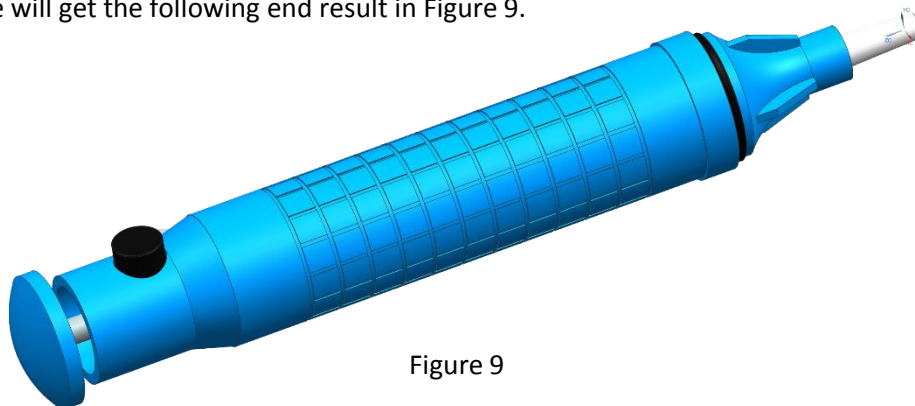


Figure 9



**Resources:**

<http://www.siemens.com/entry/cc/en/>