CITY COLLEGE

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HOMEWORK 3: Solid Modeling

Reverse Engineering Optical Microscope

ME 371 Computer Aided Design

Section: 3362

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1. Overview:

This is a multi-purpose optical microscope that can be used as a traditional microscope with powers up to 600 X or has a detached able digital camera with USB port to view the high resolution image generated by the microscope in computer and capture the snapshot or record the video as well. It's the perfect tool for discovering and learning about the world we can't see.

2. Procedure:

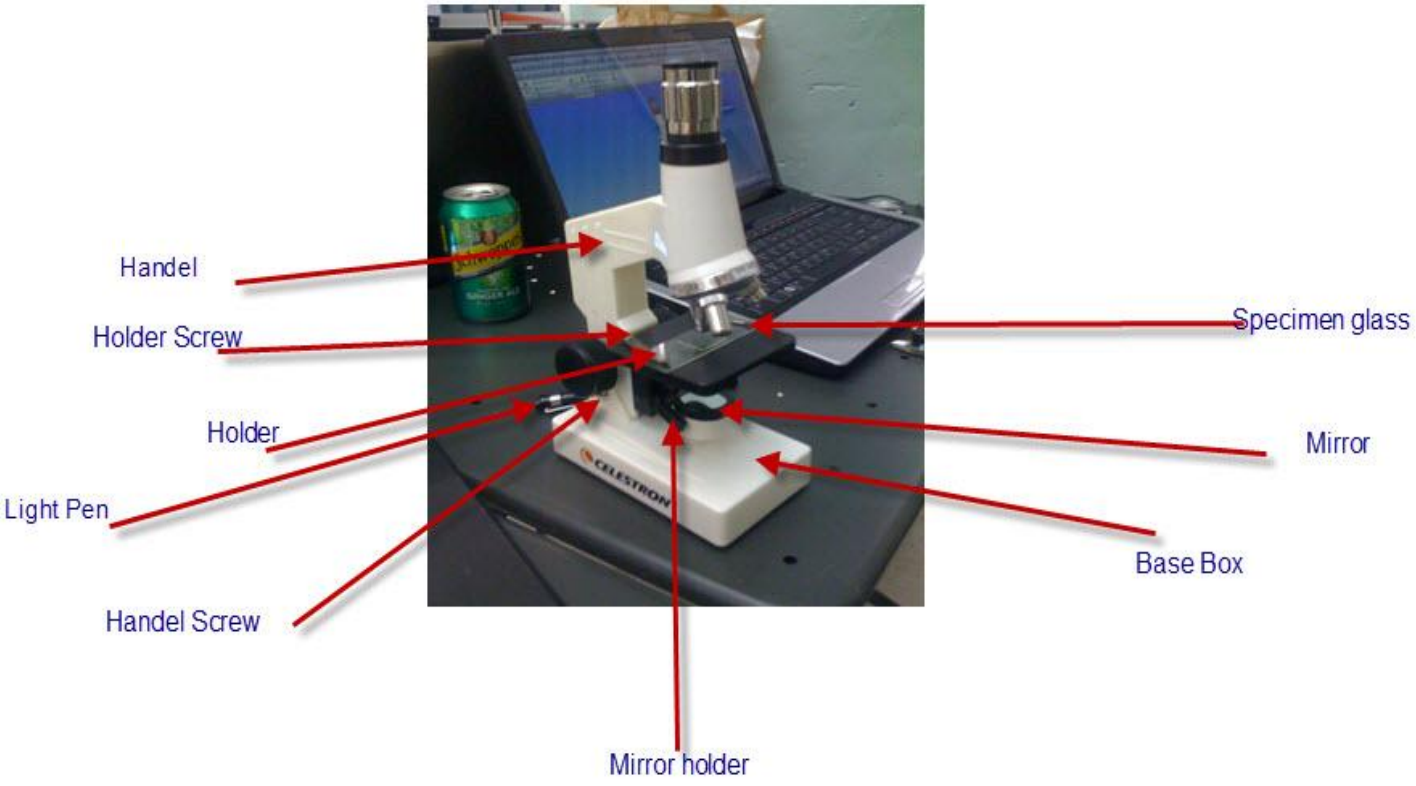


Figure 1.0 Division of Microscope into 9 Major Components

The design process involves dividing the microscope into nine components as indicated in the figure above. Then after designing the components they are finally assemble into one single device. Assemble is a phenomenal power of solidworks which has made the process of reverse engineering to its ease. The major steps in making this device in solidworks involve measuring each component using Venier Caliper and regular Hard Plastic Scale. Most of the dimensions which were linear and radial were easily measured with caliper however designing the lens and observation object piece holder part in the handle component was complex. It involves features likes draft, radial pattern, loft, fillets, chamfer and blend.

Part 1: Light Pen

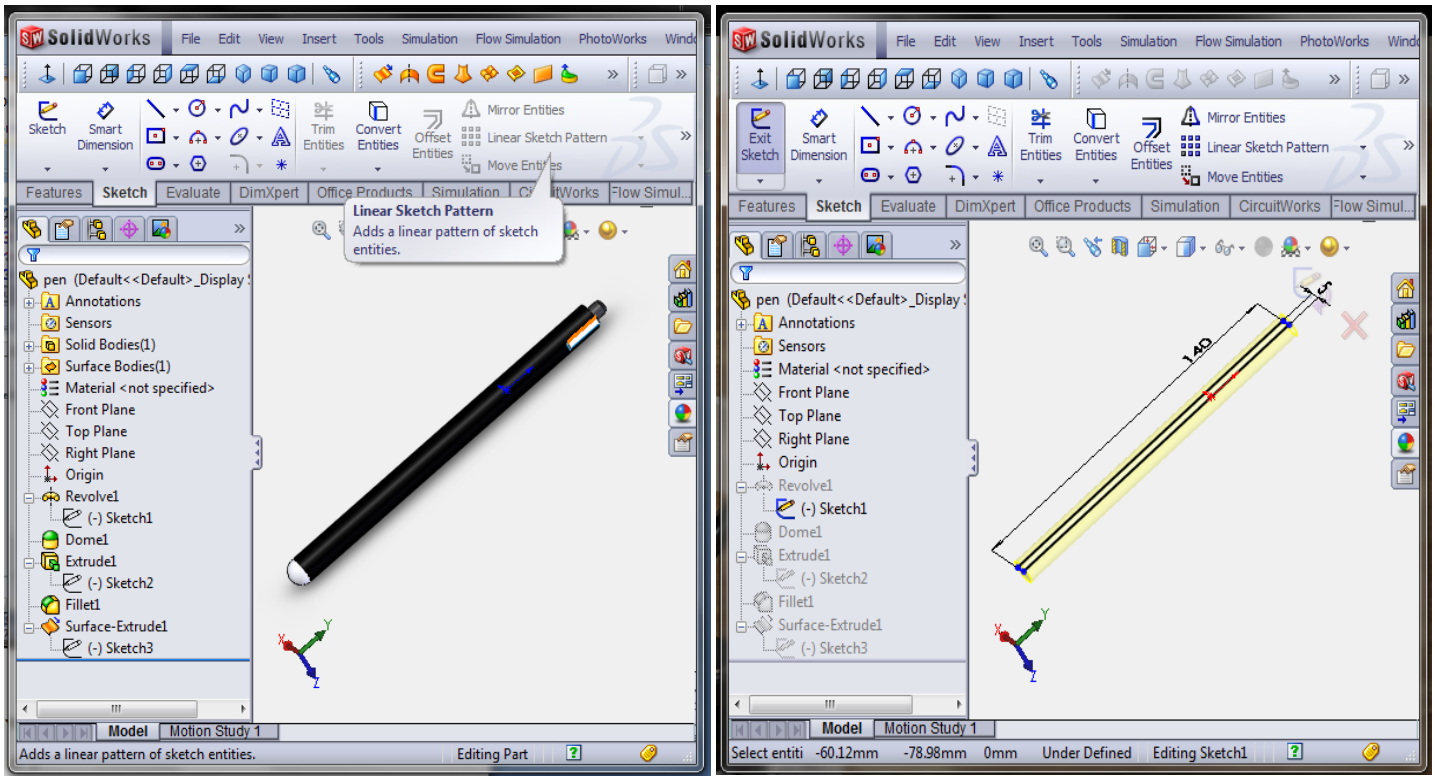


Figure 2.0

To create this component, a rectangle of length and width equals to the length of pen as measured with scale and half the diameter of the circular cross section of the pen measured with caliper respectively was assigned. Then the sketch was revolved around the longer side to generate the cylindrical body. The push switch at the top was build with extrusions of the concentric circle from the top surface. The light tip was obtained by doming the lid.

Part 2: Mirror

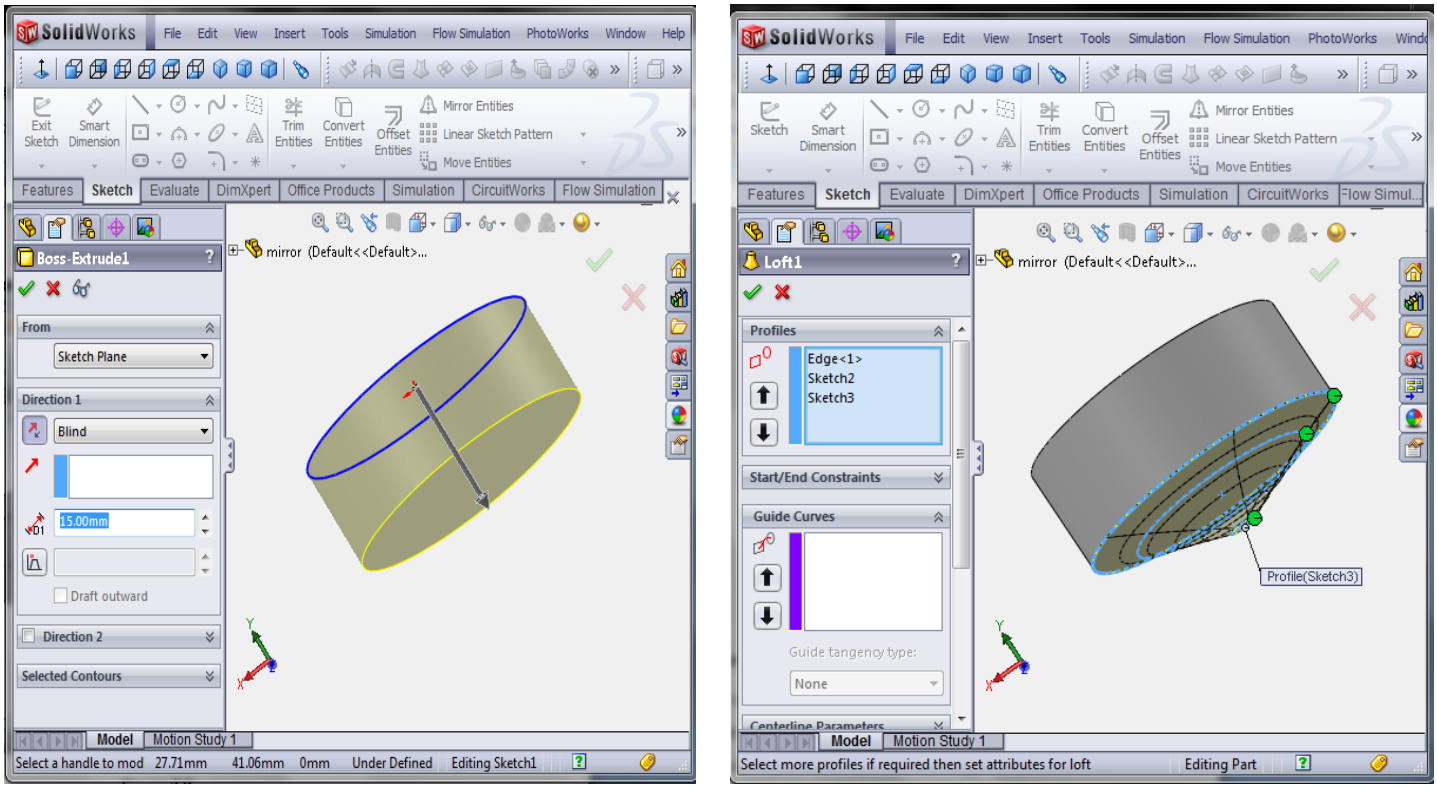


Figure 3.0

To create this component, a circle was created on the top plane and extruded measured depth. With reference to the bottom surface two reference plane were generated. With origin as center two concentric circles were created or different diameter. Then the smallest circle was lofted with reference to the medium circle on the top plane followed with big circle created on the extruded surface.

Part 3: Holder Screw and Handel Screw

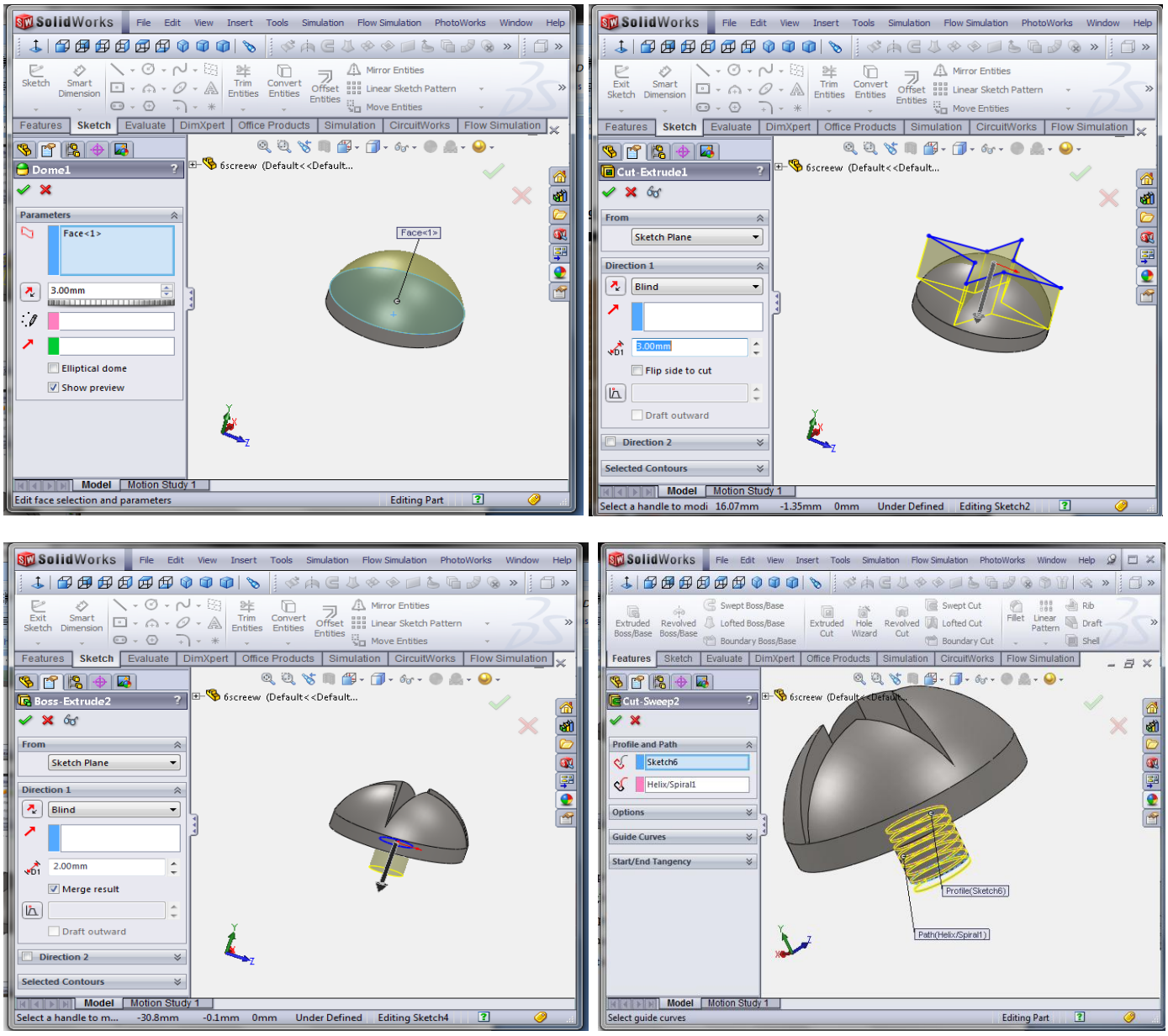


Figure 4.0

To create this component, a circle was created on the top plane and a dome was generated that is in proportional height. A reference plane was created in reference to the base of the extruded circle. A star shape was sketch on the new plane and an extruded cut was made to generate the screw head shape. On the bottom surface of the screw head a sketch of small concentric circle was created and extruded. A helix spiral curve was created on the surface on

new extruded cylinder from the bottom surface. A reference plane was generated normal to curve and a semi circle was sketched on the new plane. A loft cut was made with semicircle was primary reference and spiral curve as path to generate threads in the screw.

Part 4: Mirror holder and specimen glass holder

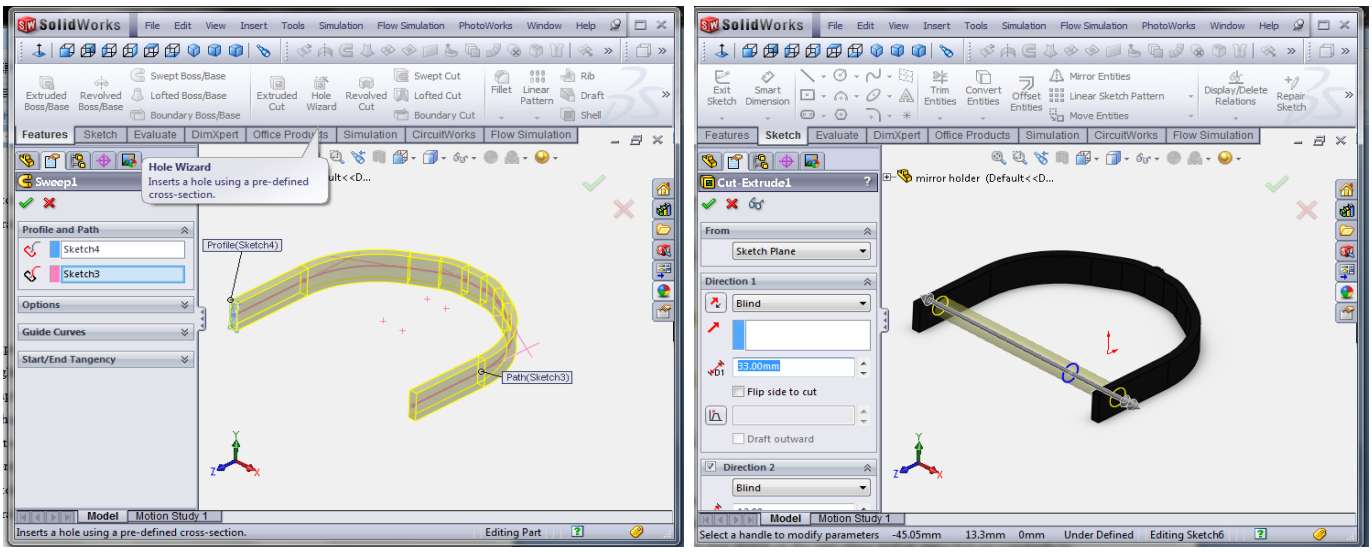
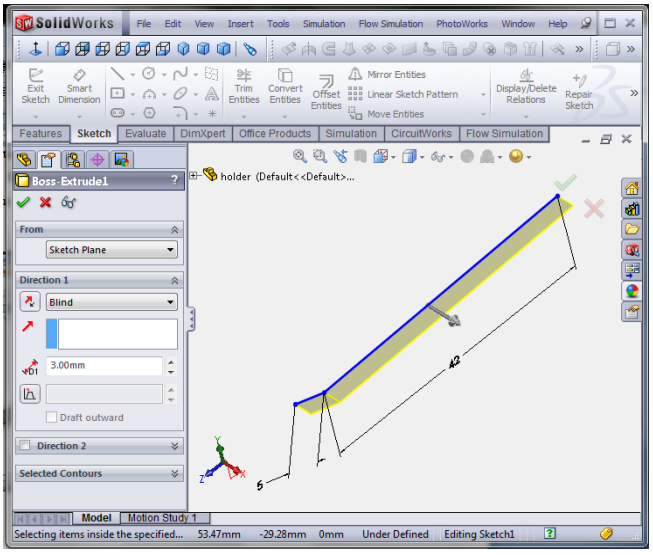


Figure 5.0

To create this component, an open 3d sketch was created on the XZ with proper required symmetric dimensions. A reference plane normal to the sketch was generated. A center rectangle was sketched with curve start point as origin. Then the rectangle was sweep with curve as reference path to generate the holder frame. A small circle was created on the right plane and extrusion cut was made out of this sketch up to the surface on the both side to generate the holder grove space.



The figure on the right is the specimen holder component which was the simplest component of all. The body was created with small extrusion of the sketch created on the right plane. An extrusion cut was done out of the circular sketch made at the end of the holder frame.

Part 5: Base box

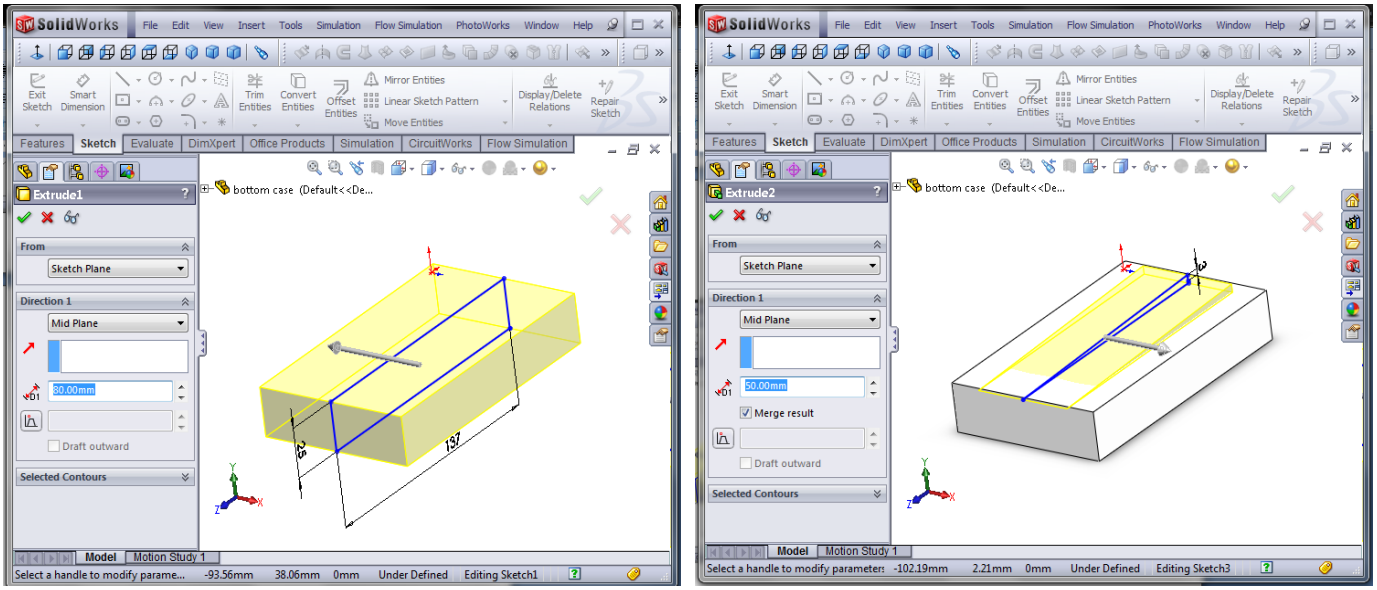


Figure 7.0

The base box component was created by extruding a rectangle and filleting all the side to obtain the curvature at the edges. A triangular extrusion was made on the right plane to generate the slope protrusion on the top surface.

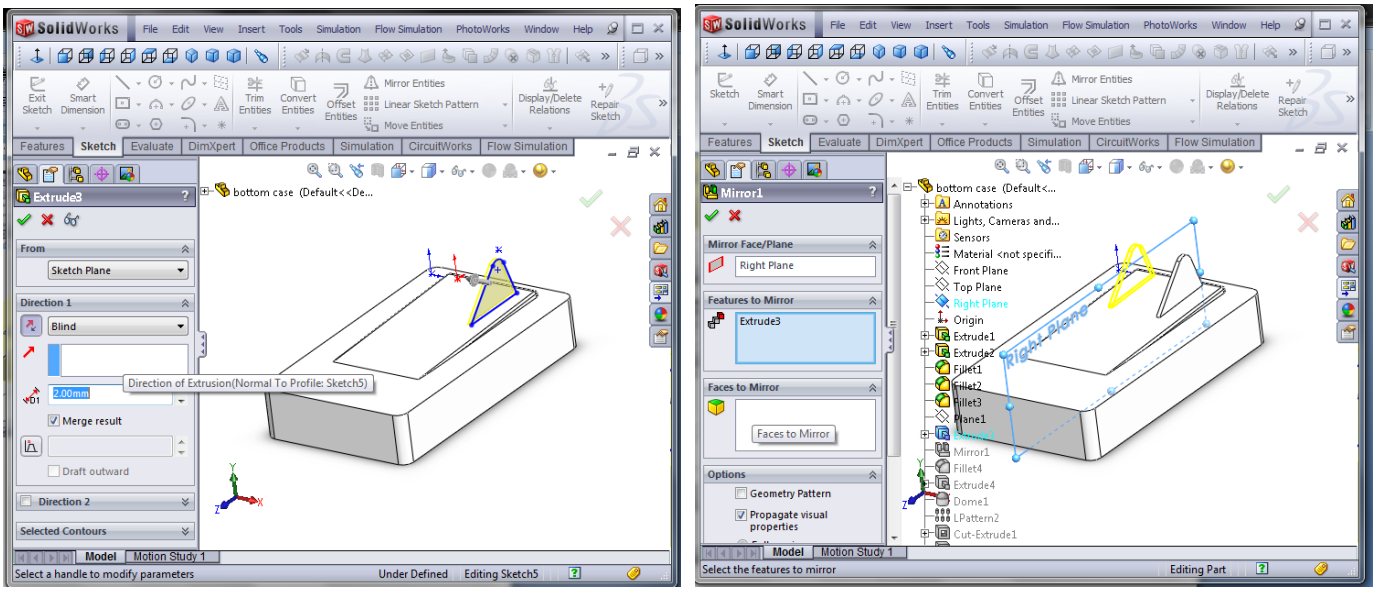


Figure 8.0

A reference plane was created with reference to right plane and a triangular sketch with round top was sketched for the holder of the handle. The extrusion was then mirror with reference to the right plane to get the holder surface on both side which is later required in assemble mates.

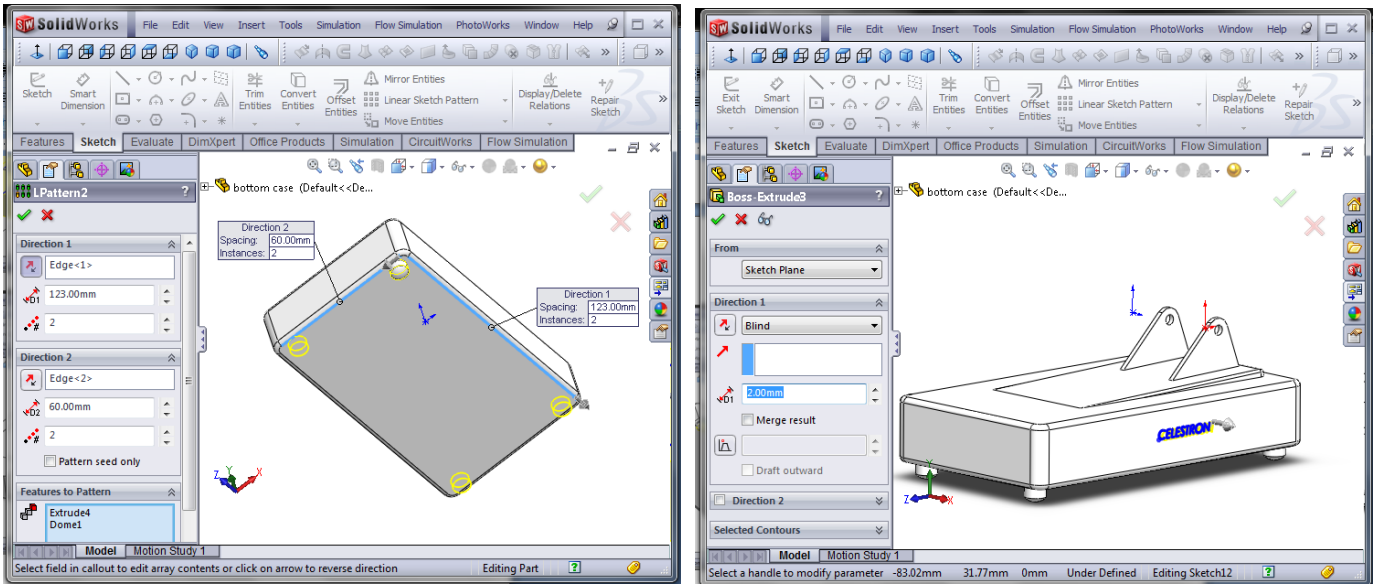


Figure 9.0

The base stand, a small cylindrical extrusion with dome bottom surface was created which was later linearly pattern in two directions with reference to two perpendicular sides to obtain all four bases in one single step. For the company logo, text extrusion was done by text sketch on the right side of the base box and later extruded to generate the contour letter.

Part6: Handel

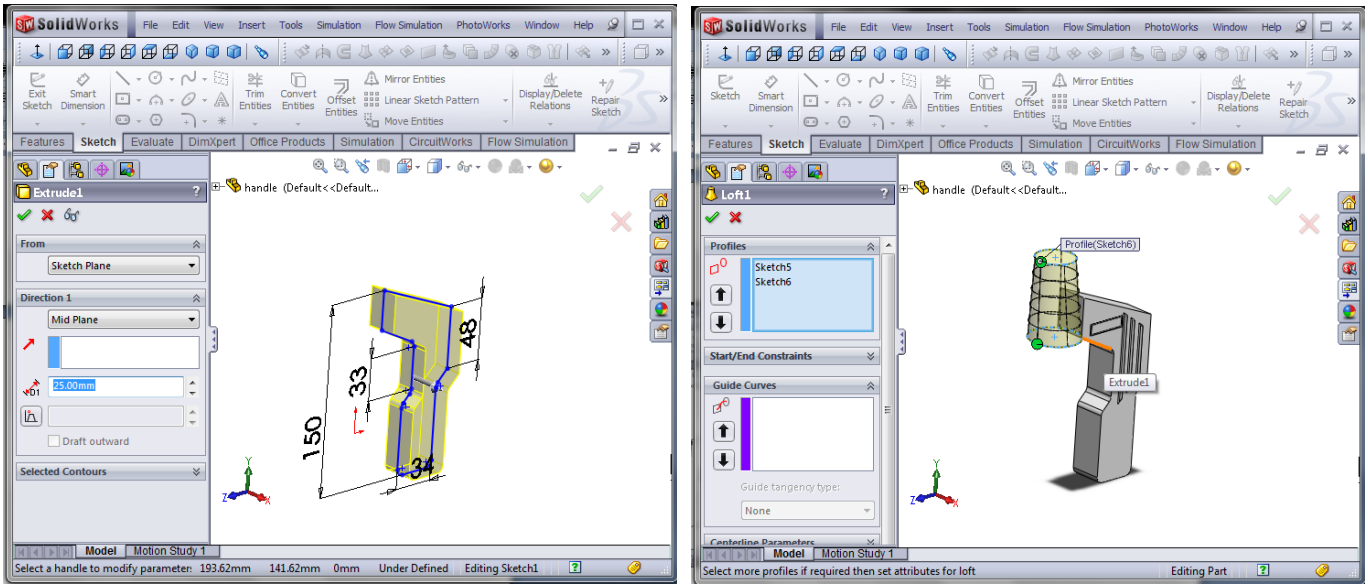


Figure 10.0

In respect to the measured dimension of the handle, a sketch was created on the right plane. The sketch was then extruded in mid plane direction to its thickness. Two reference planes was created parallel to bottom plane and a concentric circle was sketched on first reference plane and lofted to the other concentric circle sketch on top of it in another reference plane.

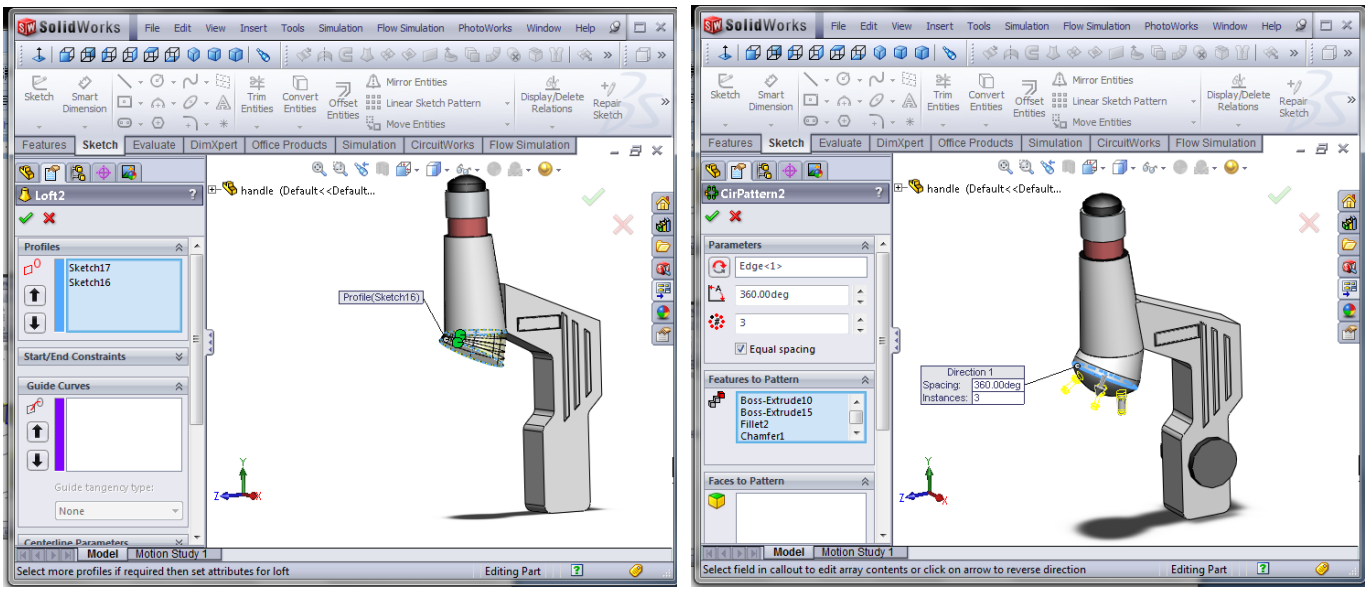


Figure 11.0

An inclined plane is created and an offset circle was sketched in this new plane. This sketch was again lofted to the surface as second reference to create a contour extrusion with curvature as desired. The three

lens fixture was created with first and cylindrical extrusion from top plane and later the features was circularly patterned along the circular cross section to generate all three lens fixture.

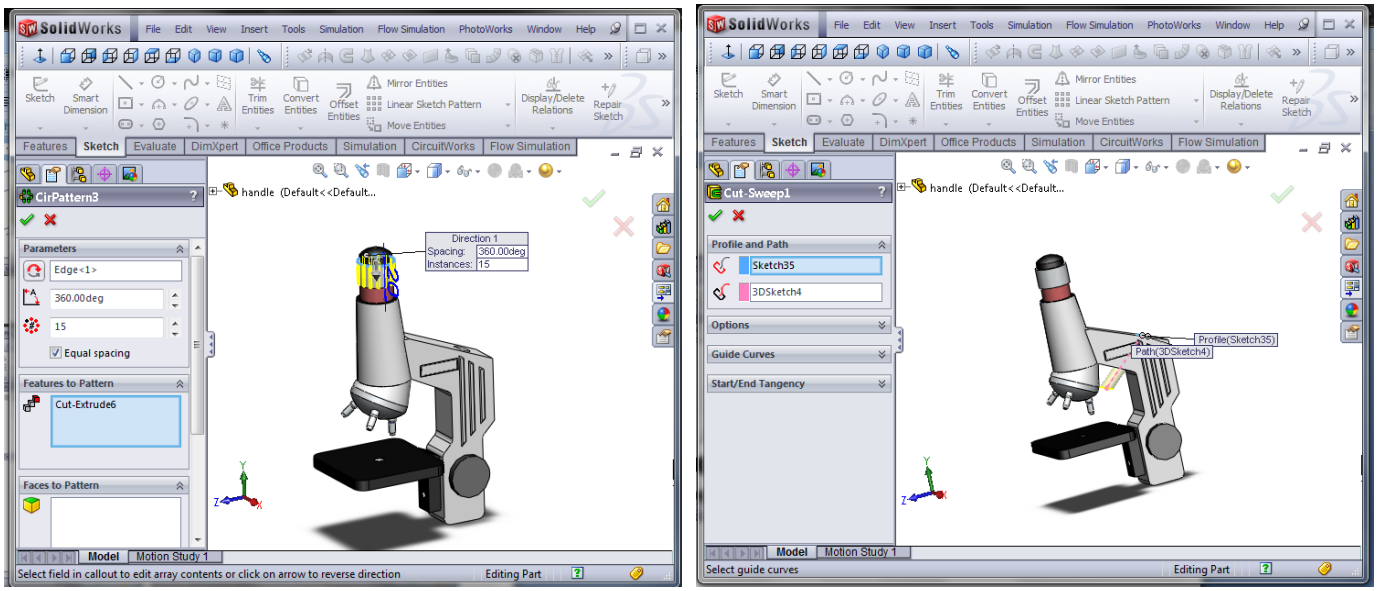


Figure 12.0

Once again circular pattern appeared hand as for the creating thread on the focus lens. A semi circular extruded cut was made first to generate first thread and later this feature pattern circular to obtain the vertical thread. For the inclined cylindrical cut and a circle was created on the profile surface and a inclined path was sketched using 3D sketch taking center of the circle as the origin of the path and extended in desired direction. This circle was then protruded as sweep cut taken 3D sketch as the path of progress, the light pen compartment was created.

3. Results:

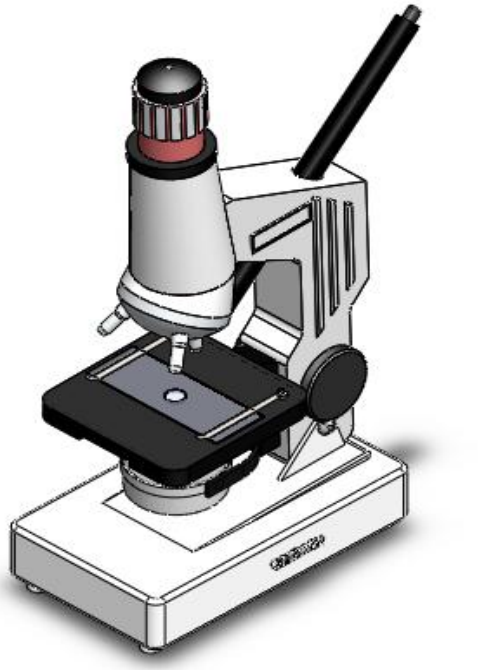


Figure 13.0 Isometric view of Assemble

Figure 14.0 Photoworks Render Image

4. Discussion:

To generate the computer model of object that I select, I divide the microscope into nine components and which was less intricate that the whole object itself. Each component was created using various features and sketch option available in solidworks. These component were later imported into a assemble file and mate together to obtain the file product model. This assignment also gave a thorough understanding of the production chain where various component are manufacture separately and assemble together to make the final product.

This assignment is one of the most riveting assignments I ever had as it evolved both the learning of reverse engineering and creative modeling of part. The microscope is aligned with precision therefore I did not disassemble it as disassembling and reassembling might disrupt the alignment and might cause focus loss. Hence most of the dimension was easily measureable without dissembling. For few cases the dimension was approximated as even the picture trace was harder to obtain. However assembling the piece might have yield more precise measurement but the approximation was also very close. Today in most industry they use high resolution 3D scanner to generate computer model of complex part which is less time consuming process and more accurate as the dimensioning is as precise as the wavelength of the scanning light.

As per the product, this microscope was made of low grade plastic which make me always cautious while using as it is more like to break. The degree of freedom for the vertical object lens focusing is not of as desired and the image quality generated in not of high grade as was in product filer. Considering these factors, the price of ninety nine dollar is not worthy to send for these products however improvement in any of these items might not only increase product life but also bring good reviews in product marketing.

Though Solidworks is one of the leading and most popular CAD software in market today, however the major issue in the solidworks is the organization of the tools and features in the window. Most of the features are not visible in the windows as one has to go to each option to find the desired function or command. Solid combine feature in solidworks is not very user friendly as in other CAD software. Solidworks should have features like one insert all six view of the part and the software should automatically generate the 3d model. Nevertheless, Solidworks in comparison to all other software I have came across including Autodesk Inventor, Solidedge and CATIA V5, is the most easy to learn, user friendly and versatile CAD software available. Within a month of a use, through the basic understand from tutorial and class lectures, creating complex geometry is just a click of the mouse away. Solidworks provides not only design, but also simulation and analysis which were however not used in the assignment, but these addins has made the potential use of this software more than any other competitive software especially in academic regime as it is very easy to learn for beginner.