

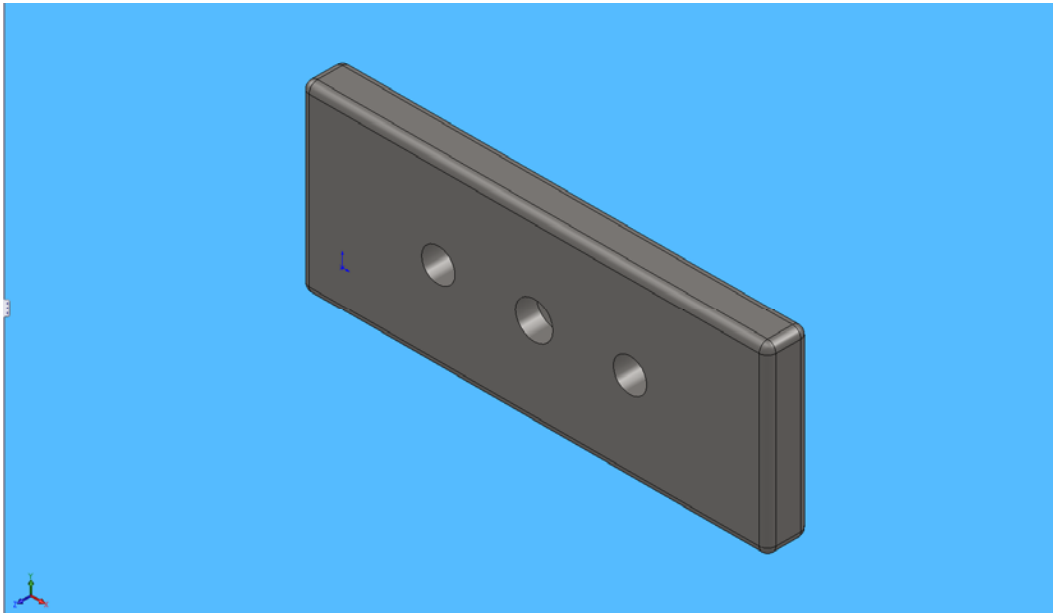
CDI CAD DOCUMENTATION PACKAGE

STUDENT NAME: ***** *****

CLASS-CDI: 60E - SOLIDWORKS - FALL 2010

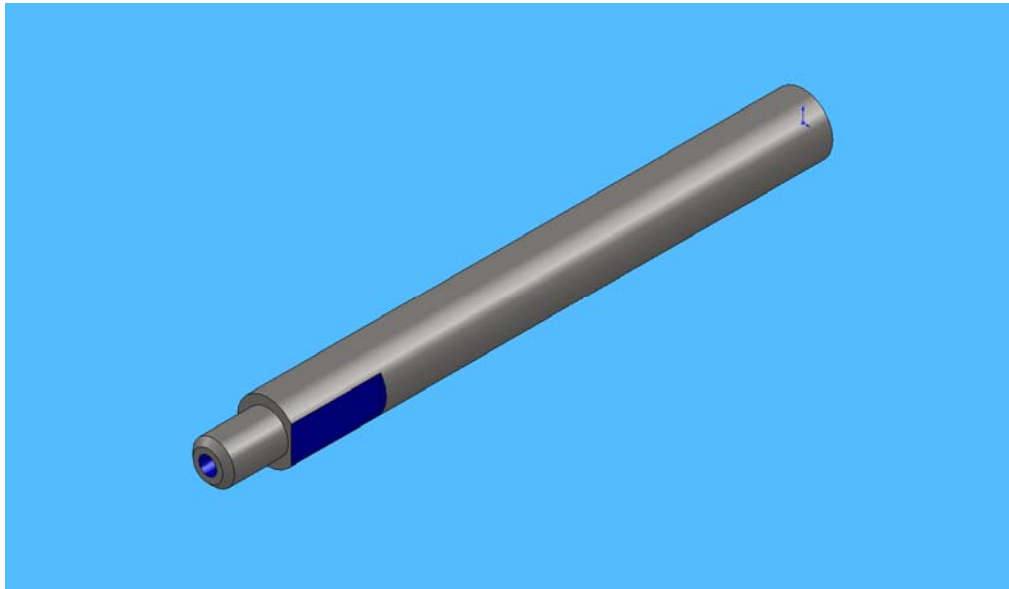
INSTRUCTOR NAME: KENNETH LOUIE

PROJECT 1/LESSON 2/PLATE



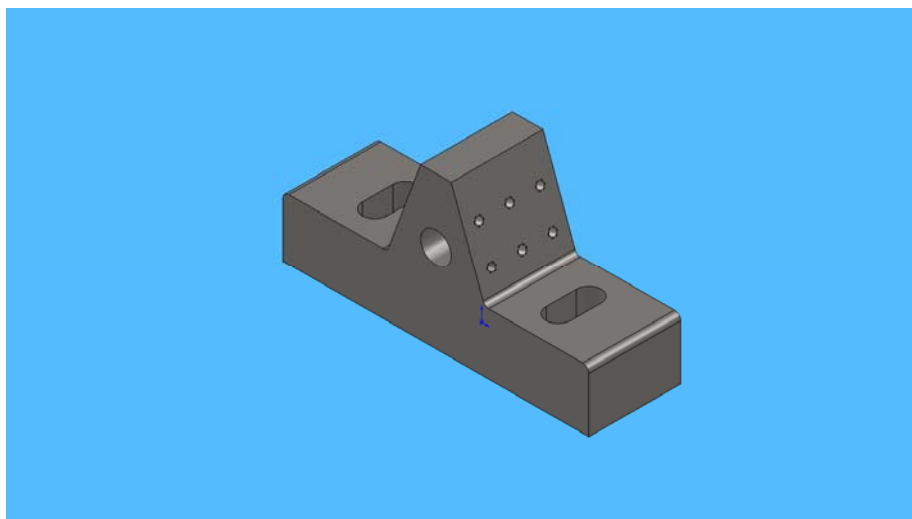
- USED SKETCH TOOLS AND PARAMETRIC FEATURES I.E. EXTRUDE BOSS/BASE , EXTRUDED CUT, HOLE FEATURE, & FILLET COMMAND TO CREATE AISI 304 STEEL PLATE.

PROJECT 1/LESSON 3/ROD



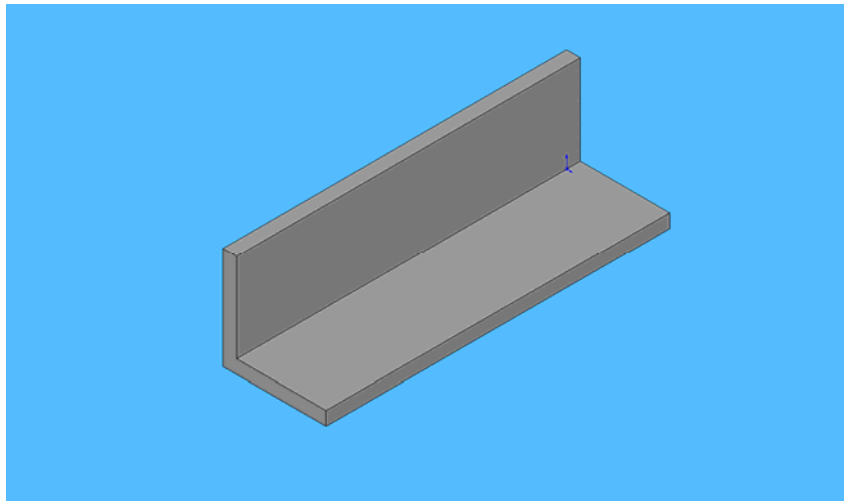
- USED SKETCH TOOLS (CONVERT ENTITIES) AND PARAMETRIC FEATURES I.E. EXTRUDE BOSS/BASE , EXTRUDED CUT, HOLE FEATURE, & CHAMFER COMMAND TO CREATE AISI 304 STEEL ROD. ADDIITIONALLY, LEARNED TO IMPLIMENT DESIGN INTENT IN MODEL, TO MAKE FOR EASE IN MODIFICATION OF ROD.

PROJECT 1/LESSON 4/GUIDE



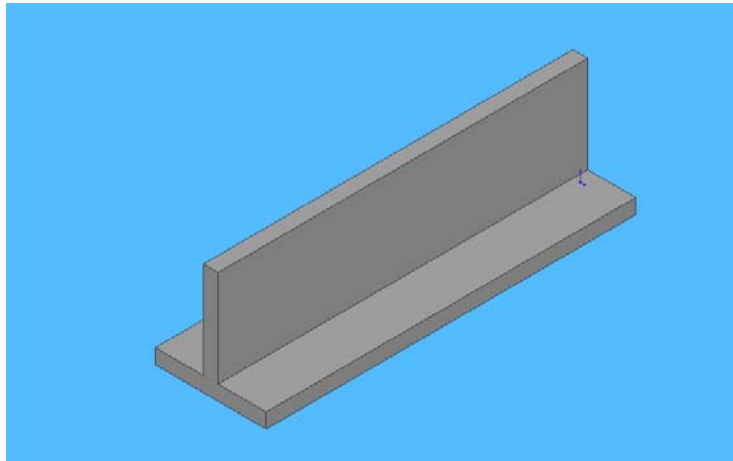
- USED SKETCH TOOLS (DYNAMIC MIRROR & SLOT) AND PARAMETRIC FEATURES I.E. EXTRUDE BOSS/BASE , EXTRUDED CUT, HOLE FEATURE, MIRROR FEATURE, AND FILLET FEATURE TO CREATE AISI 304 STEEL GUIDE. IMPLIMENTED DESIGN INTENT IN MODEL, TO MAKE FOR EASE IN MODIFICATION OF ROD.

PROJECT 1/LESSON 5/EXERCISE 1.1 L-BRACKET



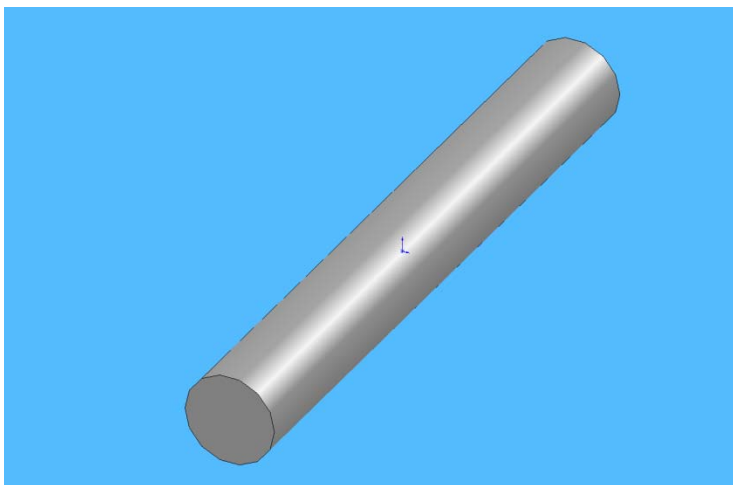
- USED SKETCH TOOLS (ADD RELATIONS) AND PARAMETRIC FEATURES I.E. EXTRUDE BOSS/BASE , TO CREATE A SIMPLE L-BRACKET.

PROJECT 1/LESSON 6/EXERCISE 1.2 T-SECTION



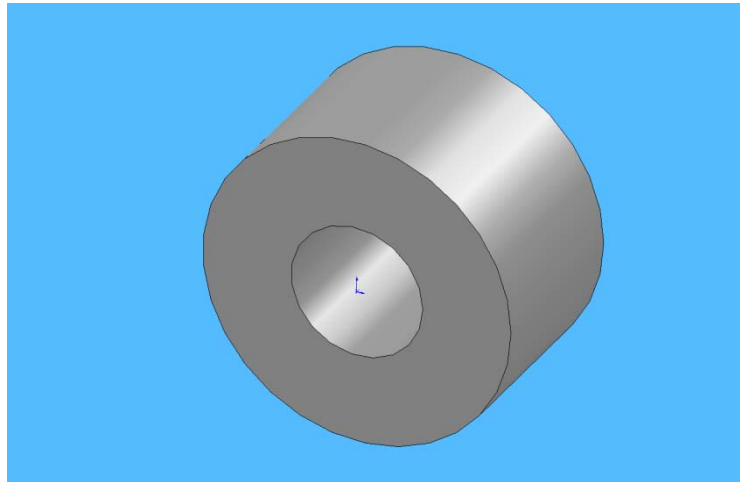
- USED SKETCH TOOLS (DYNAMIC MIRROR & ADD RELATIONS) AND PARAMETRIC FEATURES I.E. EXTRUDE BOSS/BASE , TO CREATE A SIMPLE T-SECTION

PROJECT 1/LESSON 7/EXERCISE 1.8 AXLE



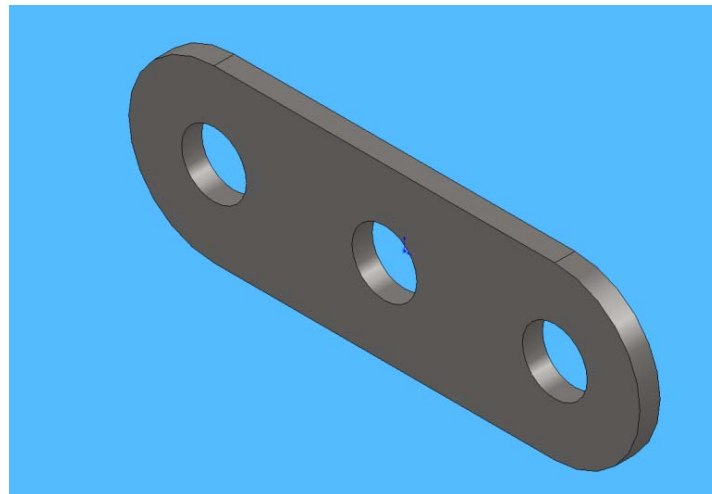
- USED SKETCH TOOLS AND PARAMETRIC FEATURES I.E. EXTRUDE BOSS/BASE , TO CREATE THE AXLE CYLINER.

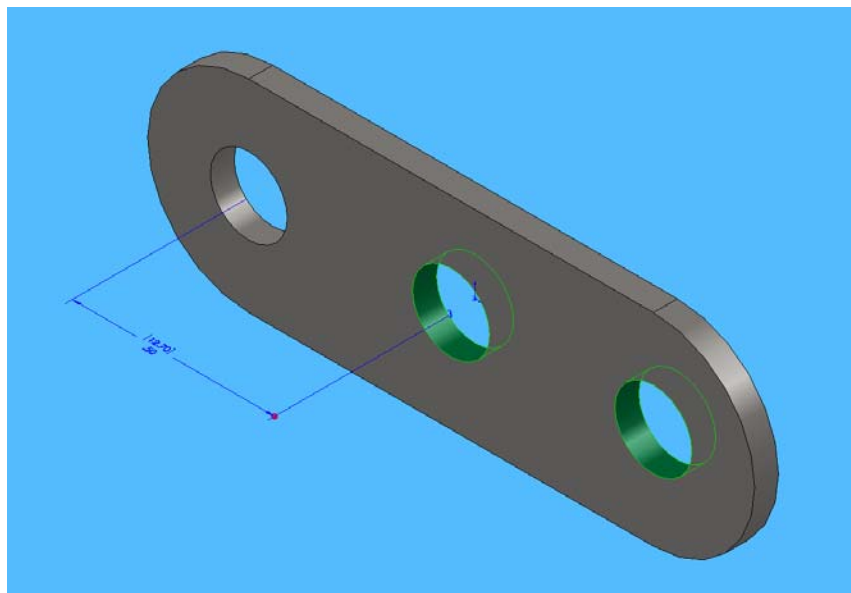
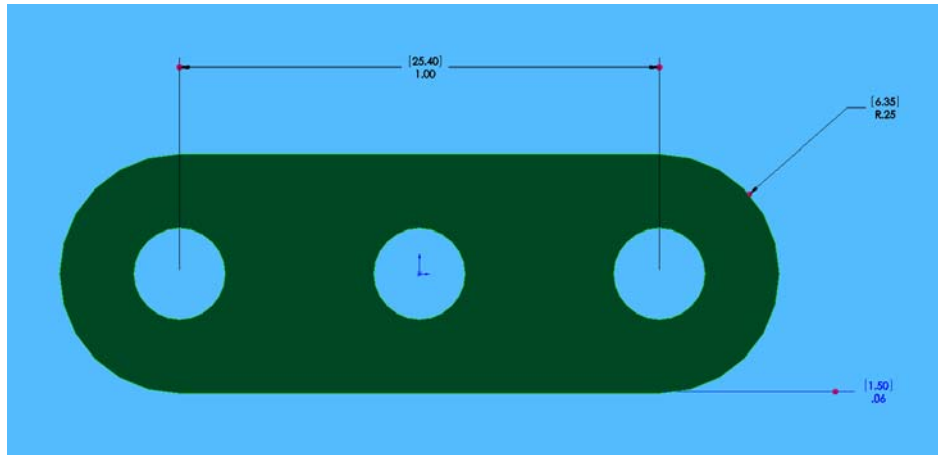
PROJECT 1/LESSON 8/EXERCISE 1.9 SHAFT COLLAR



- USED SKETCH TOOLS AND PARAMETRIC FEATURES I.E. EXTRUDE BOSS/BASE AND EXTUDE CUT TO CREATE THE SHAFT COLLAR.

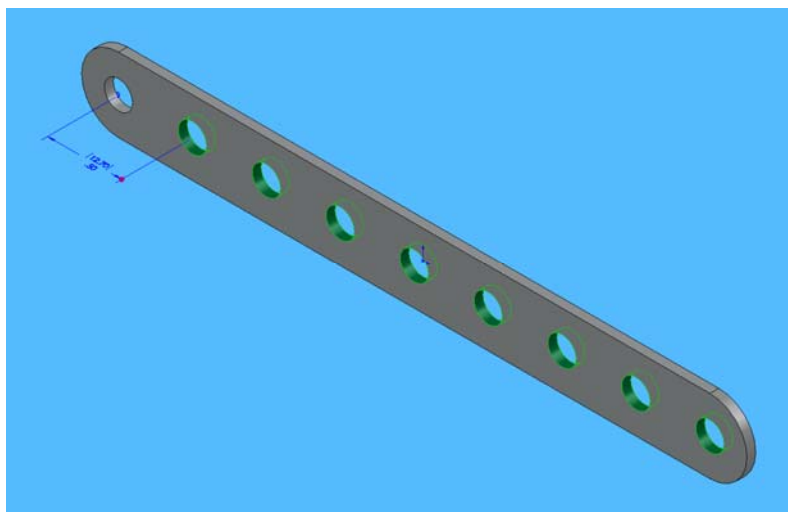
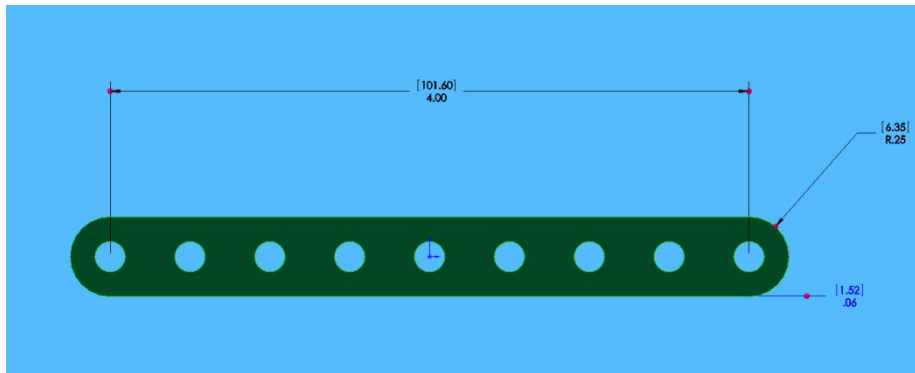
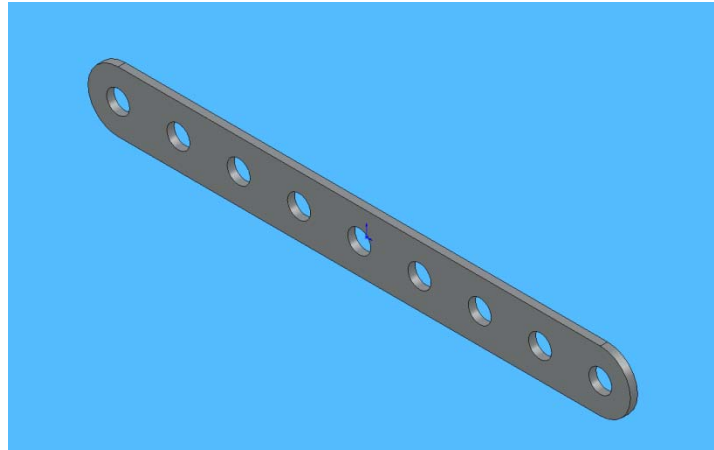
PROJECT 1/LESSON 9/EXERCISE 1.10 FLAT BAR (3 HOLES)





- USED SKETCH TOOLS (SLOT, DUAL DIMENSIONS) AND PARAMETRIC FEATURES I.E. EXTRUDE BOSS/BASE , EXTUDE CUT & LINEAR PATTERN TO CREATE THE FLAT BAR (3 HOLES) PART.

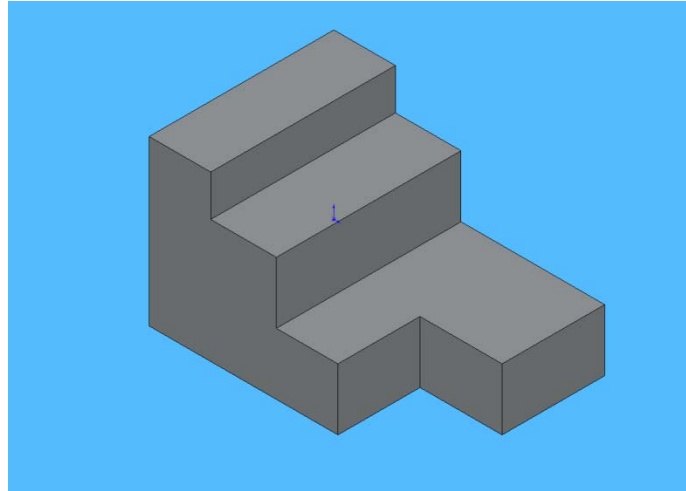
PROJECT 1/LESSON 10/EXERCISE 1.11 FLAT BAR (9 HOLES)



- MODIFIED THE FLAT BAR (3 HOLES) PART BY CHANGING THE THE LENGTH DIMENSIONS, AND INCREASING THE NUMBER OF HOLES

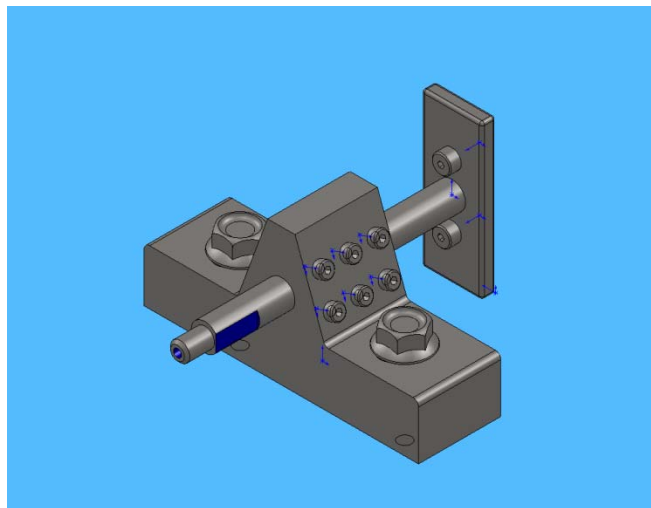
TO PATTERN IN ORDER TO CREATE THE FLAT BAR (9 HOLES)
PART.

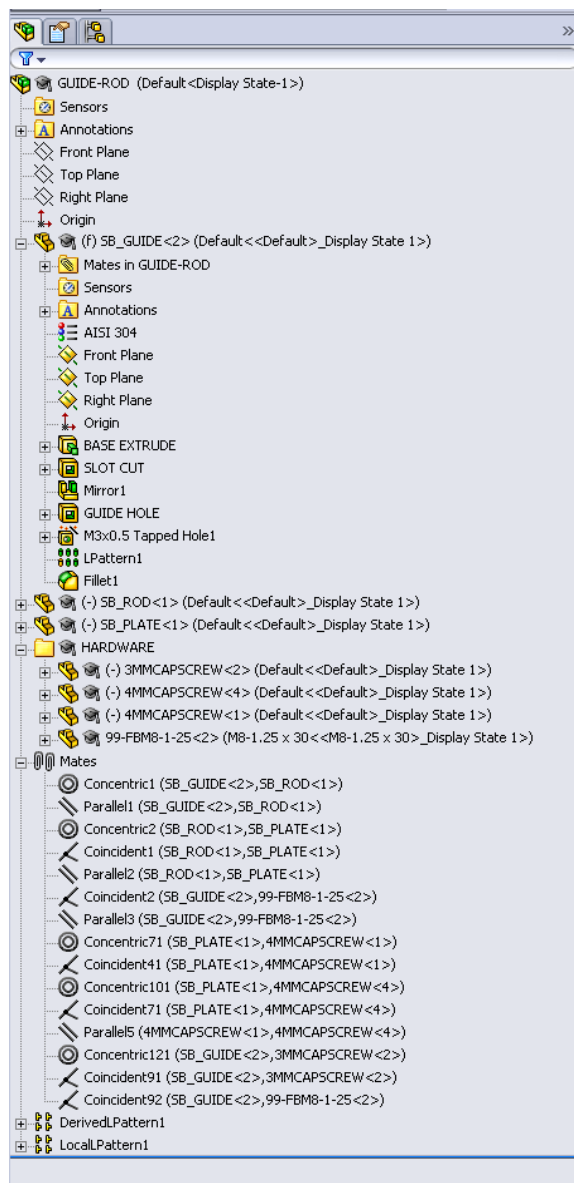
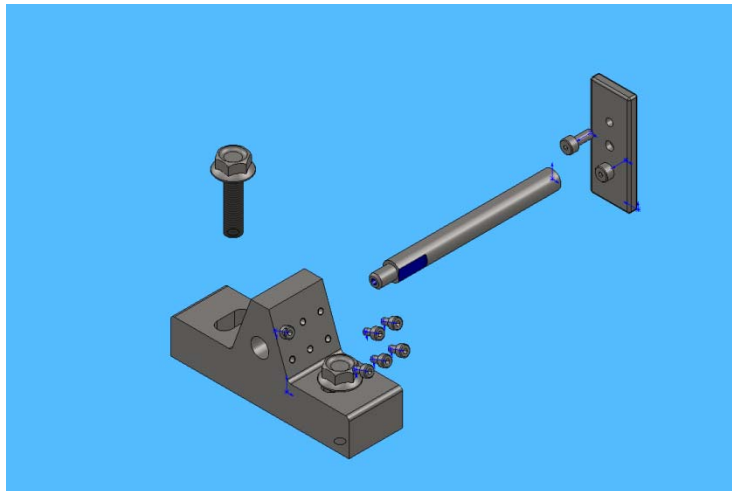
PROJECT 1/LESSON 11/EXERCISE 1.14 STEP BLOCK



- USED SKETCH TOOLS AND PARAMETRIC FEATURES I.E. EXTRUDE BOSS/BASE AND EXTRUDED CUT, TO CREATE 2014 ALLOY STEP BLOCK.

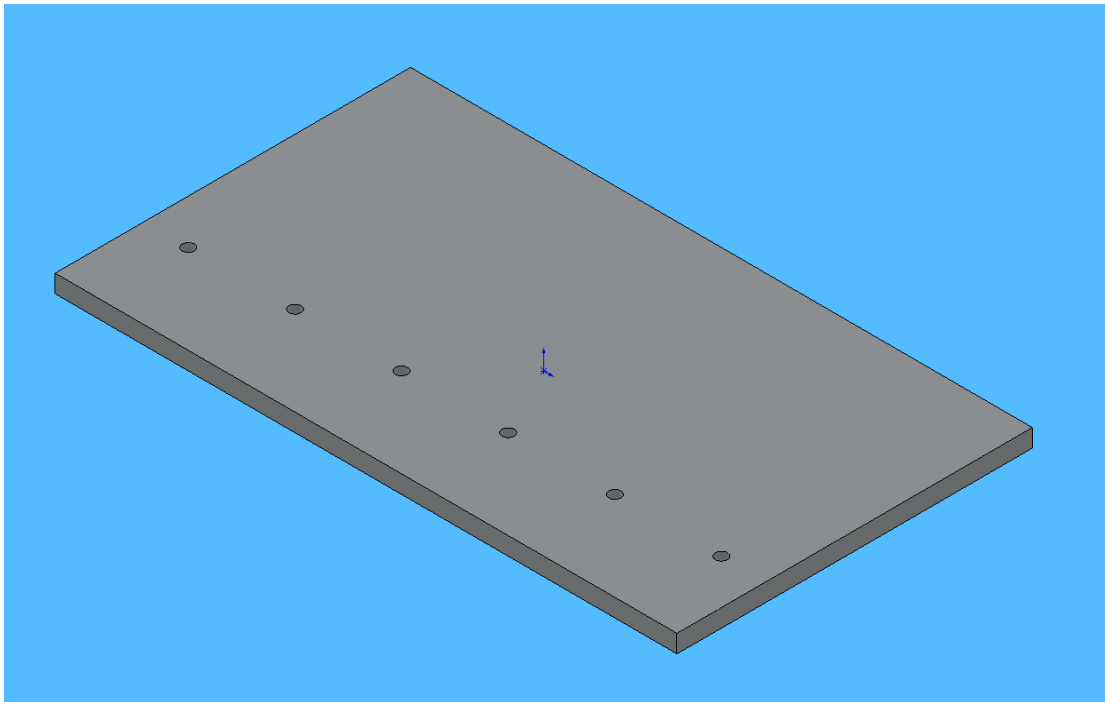
PROJECT 2/LESSON 12/GUIDE, ROD, PLATE AND ALL

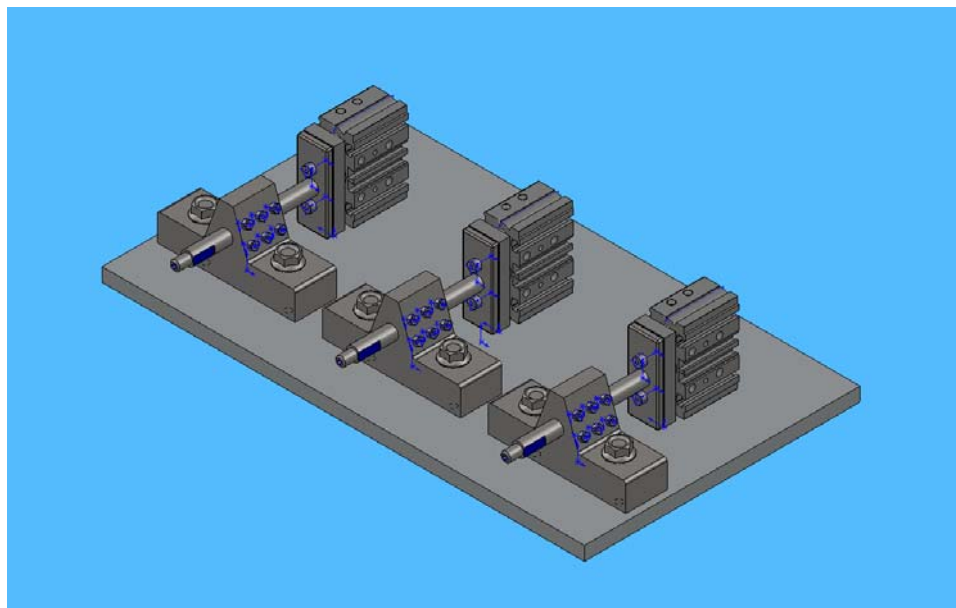
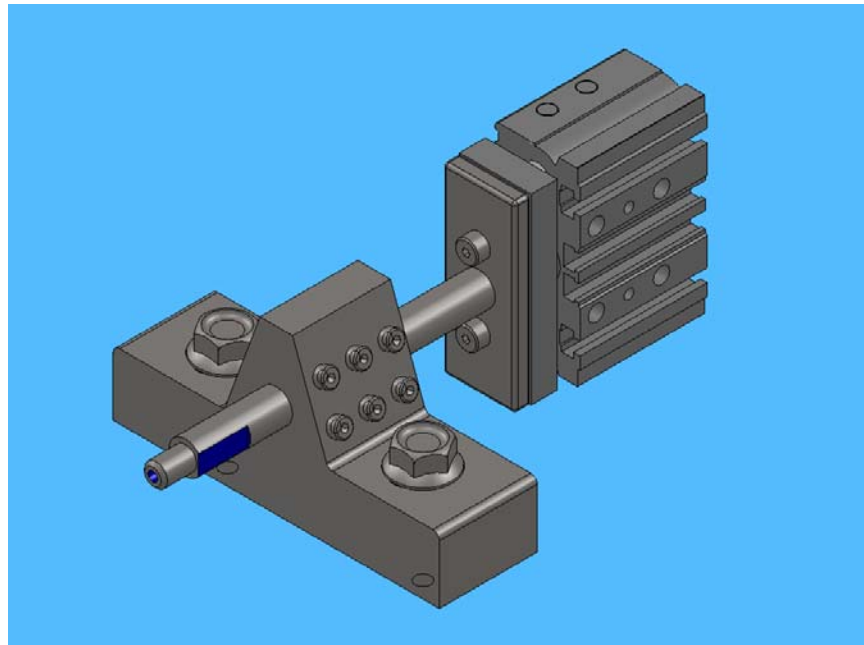




- CREATED AN ASSEMBLY FOR THE GUIDE, ROD, AND PLATE. IMPLIMENTED SEVERAL MATING FEATURE (I.E. COINCIDENT, CONCENTRIC & PARALLEL MATES) TO CONSTRAIN FEATURES. INCLUDED FEATURES FROM THE DESIGN LIBRARY (FLANGE BOLT), USED THE REVOLVE FEATURE TO CREATE 4MM SCREW. USED THE COPY FEATURE TO SAVE, MODIFY AND CREATE THE 3MM SCREW. CREATED A HARDWARE FOLDER, USED THE LINEAR PATTERN AND DISPLAY PATTERN FEATURE TO MATE THE SCREW AND FLANGE BOLT TO THE GUIDE. CREATED AN EXPLODED VIEW OF THE ASSEMBLY.

PROJECT 2/LESSON 13/EXERCISE 2.4: BASE PLATE ASSEMBLY





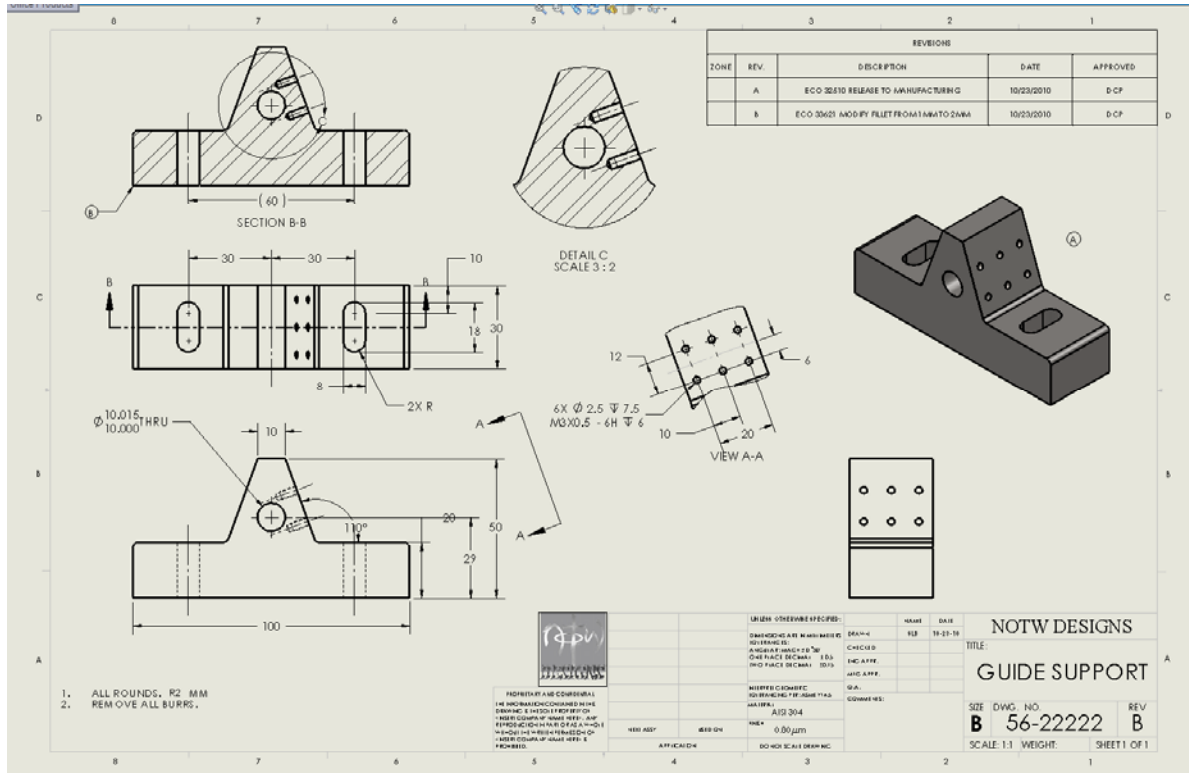
- CREATED A BASE PLATE THAT WOULD HOLD THREE CUSTOMER ASSEMBLY FIXTURES. THE CUSTOMER ASSEMBLY CONSISTED OF THE GUIDE, ROD, AND PLATE ASSEMBLY AS WELL AS A DOWNLOADED ASSEMBLY FILE, MGPM COMPONENTED. CREATED A LINEAR HOLE PATTERNS IN THE PLATE, ADDED A DERRIVED PATTERN FEATURE TO LINK THE CUSTOMER ASSEMBLY TO THE HOLE PATTERN IN THE BASE PLATE.

PROJECT 2/LESSON 14/EXERCISE 2.6: LINKAGE ASSEMBLY



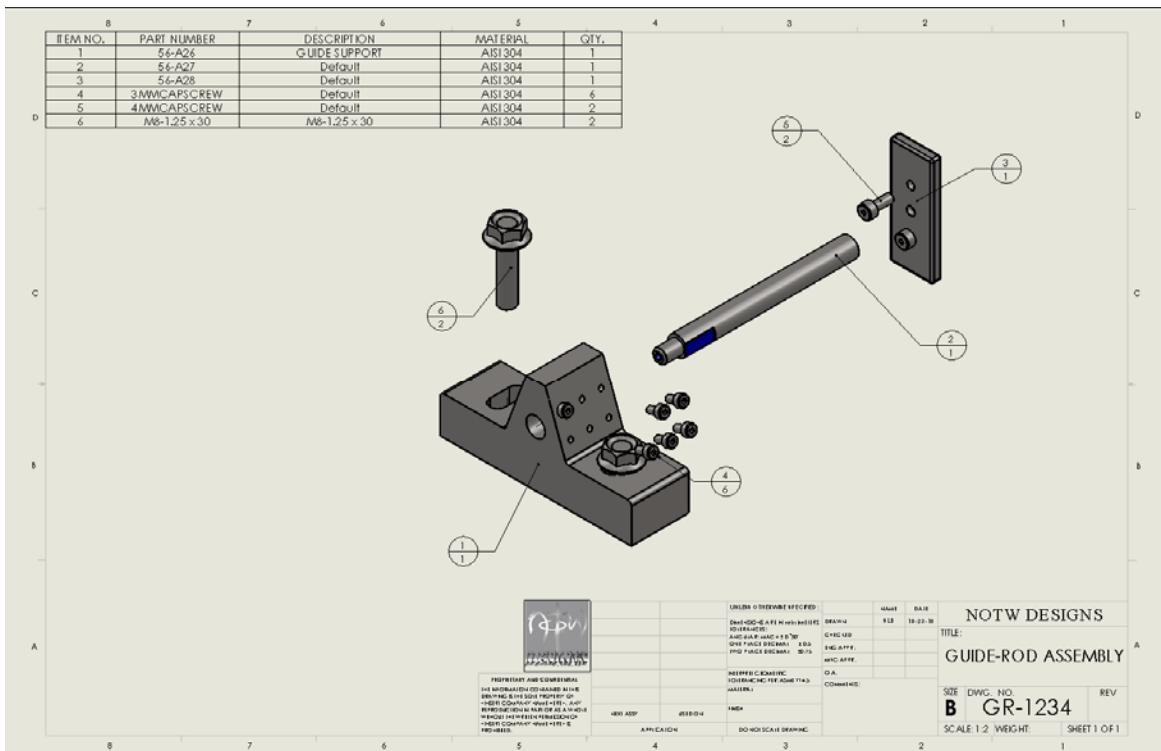
- DOWNLOADED AIR CYLINDER AND IMPORTED IT INTO THE ASSEMBLY. ADDED THE AXLE, SHAFT COLLAR, FLAT BAR (3 HOLE) & FLAT BAR (9 HOLE). IMPLIMENTED SEVERAL MATING FEATURE (I.E. COINCIDENT, CONCENTRIC & PARALLEL MATES) TO CONSTRAIN FEATURES.

PROJECT 3/LESSON 15/GUIDE PART DETAIL



- CREATED A DRAWING FROM THE GUIDE PART. MODIFIED THE DRAWING TEMPLATE TO INCLUDE PROPER UNITS (I.E. MM OR INCH DEPENDING ON THE APPLICATION) CREATED A COMPANY LOGO, LINKED FEATURES INTO THE TITLE BLOCK (I.E. COMPANY NAME, PART DESCRIPTION, PART NUMBER, DRAWN BY, DRAWN DATE AND MATERIAL). USED VIEW PALLETE TO BRING IN VIEWS TO THE DRAWING TEMPLATE. USED THE VIEW LAYOUT TAB TO APPLY SEVERAL VIEWS (I.E. SECTION, DETAIL, AND AUXILLARY VIEWS). USING THE ANNOTATION TAB AND MODEL ITEMS, DIMENSIONS WERE ADDED TO EACH VIEW. LASTELY, A REVISION TABLE WAS ADDED TO THE DRAWING WHICH PROVIDES INFORMATION ON REVISION NUMBER, DESCRIPTION OF THE PART, DATE OF REVISION, AND APPROVAL INFORMATION. LASTLY ADDED A NOTE, AND LINKED THE MATERIAL PROPERTIES TO THE NOTE

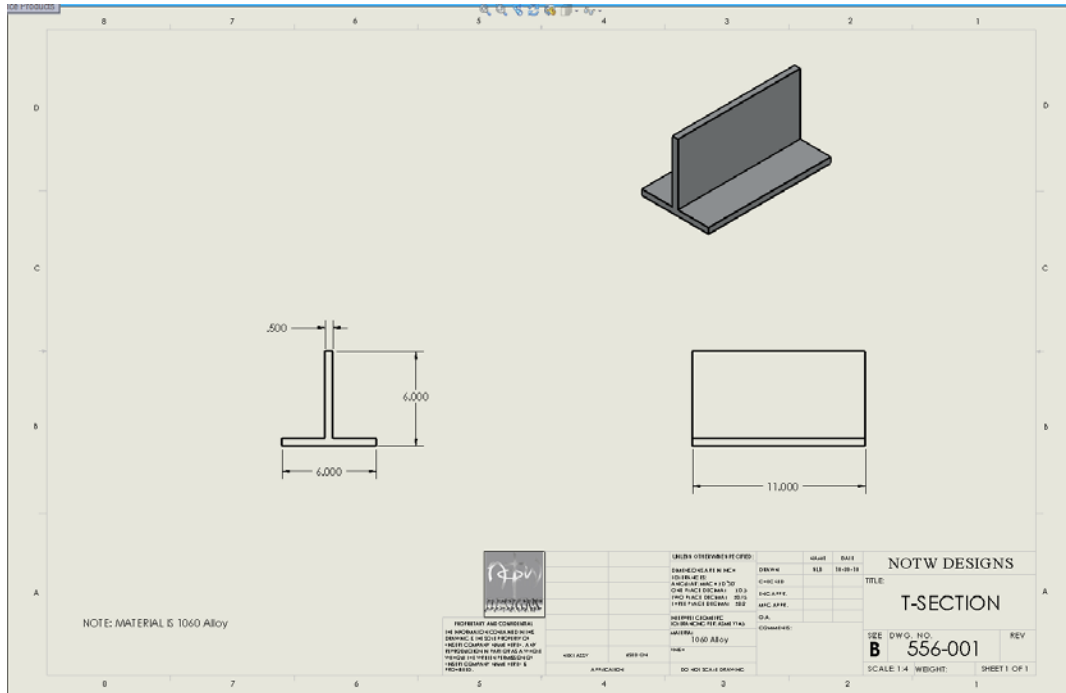
PROJECT 3/LESSON 16/GUIDE ASSEMBLY



- CREATED A DRAWING FROM THE GUIDE, ROD, PLATE ASSEMBLY. MODIFIED THE DRAWING TEMPLATE TO INCLUDE PROPER UNITS (I.E. MM OR INCH DEPENDING ON THE APPLICATION) INCLUDED COMPANY LOGO, LINKED FEATURES INTO THE TITLE BLOCK (I.E. COMPANY NAME, PART DESCRIPTION, PART NUMBER, DRAWN BY, DRAWN DATE AND MATERIAL). USED MODEL VIEW LOCATED IN THE VIEW LAYOUT TAB TO BRING THE ISO-VIEW INTO THE DRAWING TEMPLATE. USED THE ANNOTATION TAB TO INCLUDE BALLONS TO LABEL EACH COMPONENT. LASTELY, A BILL OF MATERIAL TABLE WAS ADDED TO THE DRAWING WHICH PROVIDES INFORMATION ON PART NUMBER, DESCRIPTION OF THE PART, MATERIAL PROPERTIES, AND QUANTITY OF COMPONENTS LISTED.

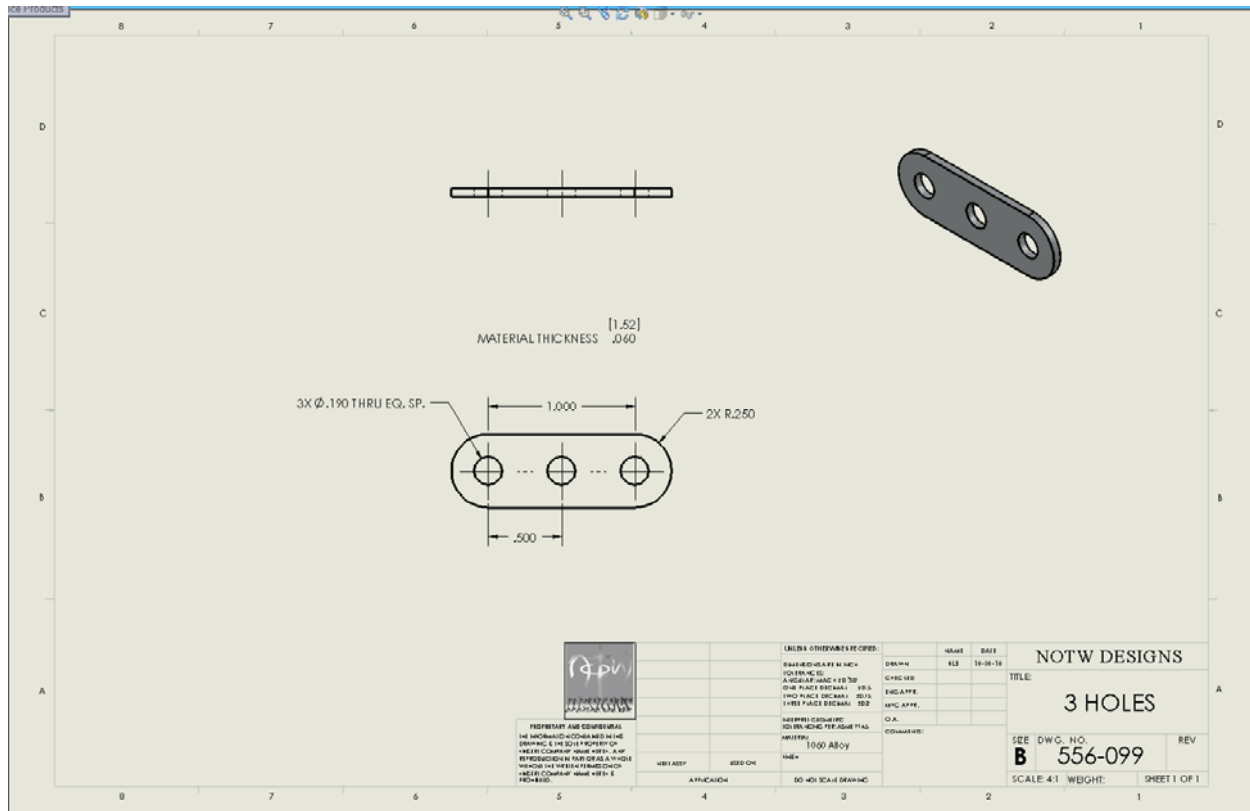
DIMENSIONS TO EACH VIEW. LASTLY, ADDED A NOTE, AND LINKED THE MATERIAL PROPERTIES TO THE NOTE

PROJECT 3/LESSON 18/EXERCISE 3.2: T-SECTION DRAWING



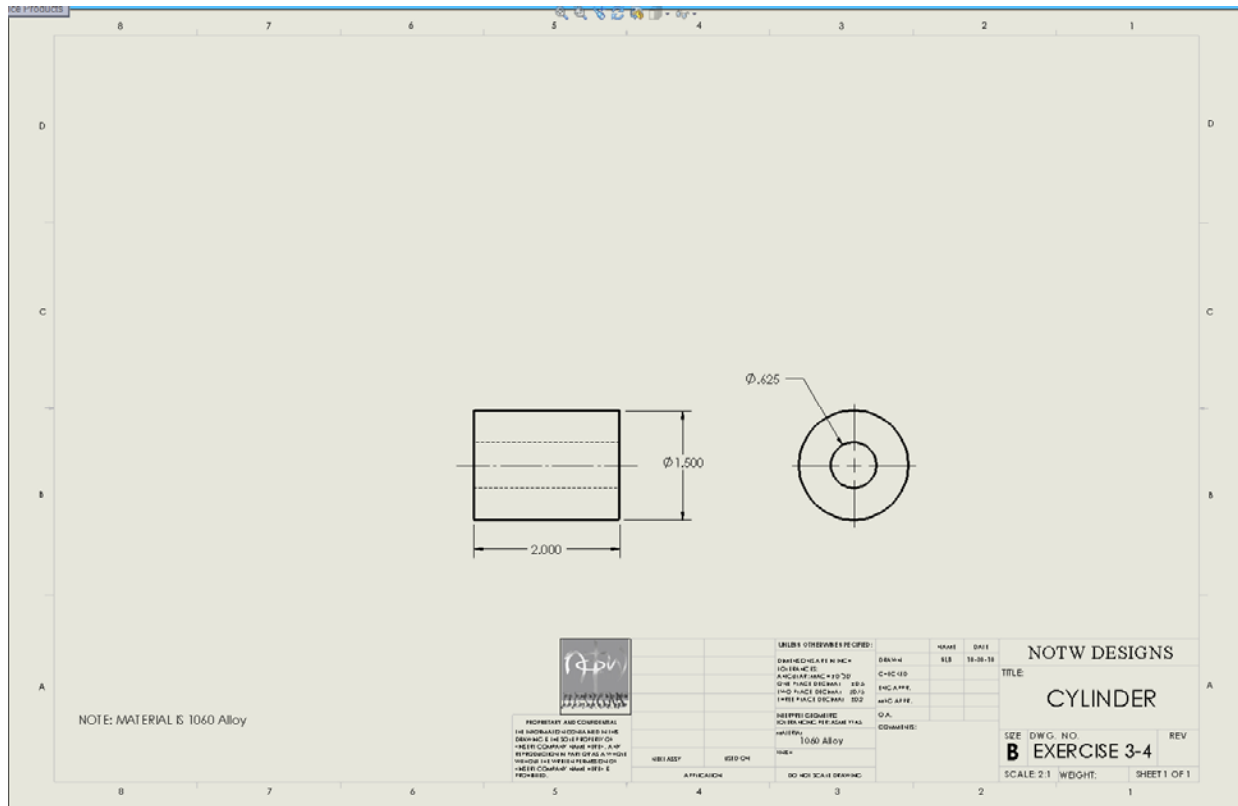
- MODELED THE T-SECTION AND CREATED A DRAWING FROM THE PART. MODIFIED THE DRAWING TEMPLATE TO INCLUDE PROPER UNITS (I.E. MM OR INCH DEPENDING ON THE APPLICATION) CREATED A COMPANY LOGO, LINKED FEATURES INTO THE TITLE BLOCK (I.E. COMPANY NAME, PART DESCRIPTION, PART NUMBER, DRAWN BY, DRAWN DATE AND MATERIAL). USED VIEW PALLETE TO BRING IN VIEWS TO THE DRAWING TEMPLATE. INCORPORATED THE LAYER COMMAND TO HIDE DIMENSIONS. LASTLY ADDED A NOTE, AND LINKED THE MATERIAL PROPERTIES TO THE NOTE.

PROJECT 3/LESSON 19/EXERCISE 3.3: FLAT BAR (3 HOLE) DRAWING



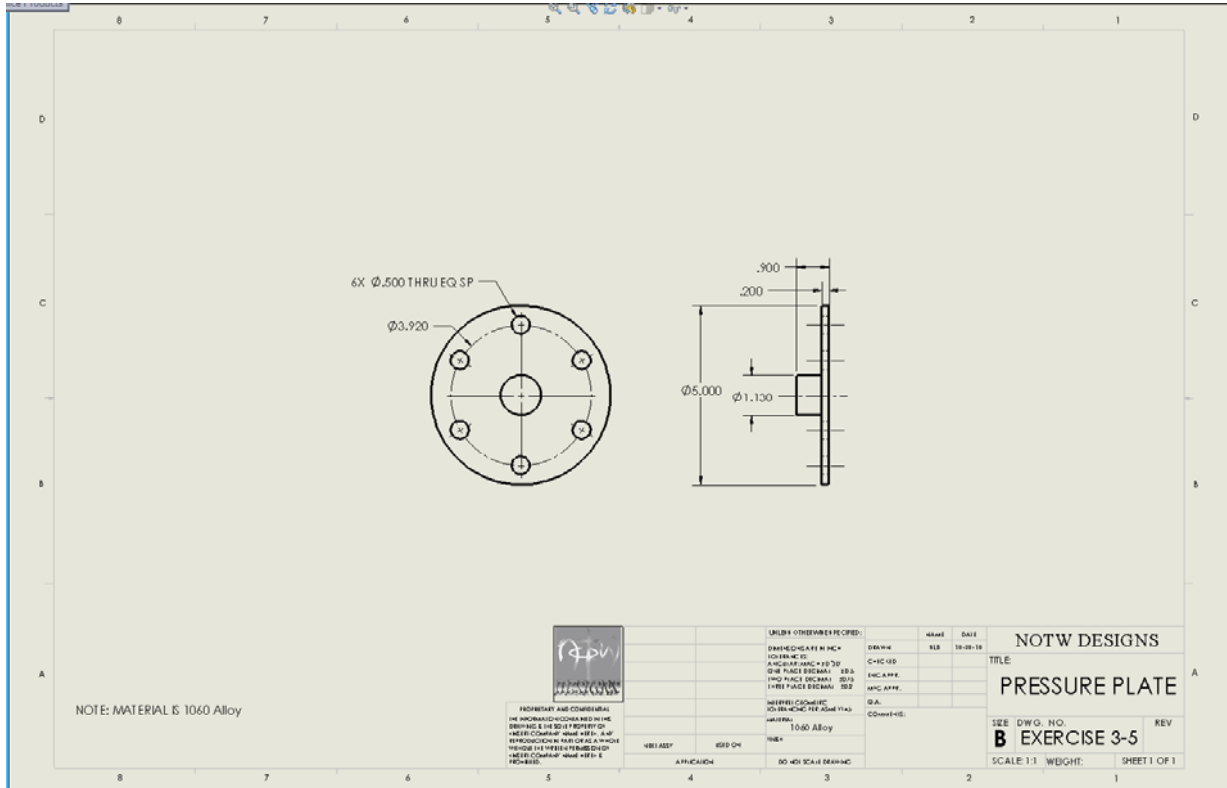
- MODELED THE FLAT BAR (3 HOLE) AND CREATED A DRAWING FROM THE PART. MODIFIED THE DRAWING TEMPLATE TO INCLUDE PROPER UNITS (I.E. MM OR INCH DEPENDING ON THE APPLICATION) CREATED A COMPANY LOGO, LINKED FEATURES INTO THE TITLE BLOCK (I.E. COMPANY NAME, PART DESCRIPTION, PART NUMBER, DRAWN BY, DRAWN DATE AND MATERIAL). USED VIEW PALLETTE TO BRING IN VIEWS TO THE DRAWING TEMPLATE. ADDED THE CENTER LINES AND CENTER MARK TOOLS IN THE ANNOTATION TAB, TO THE 3X .190 DIAMETER HOLES ON THE PART. LASTLY, INCORPORATED THE LAYER COMMAND TO HIDE DIMENSIONS.

PROJECT 3/LESSON 20/EXERCISE 3.4: CYLINDER DRAWING



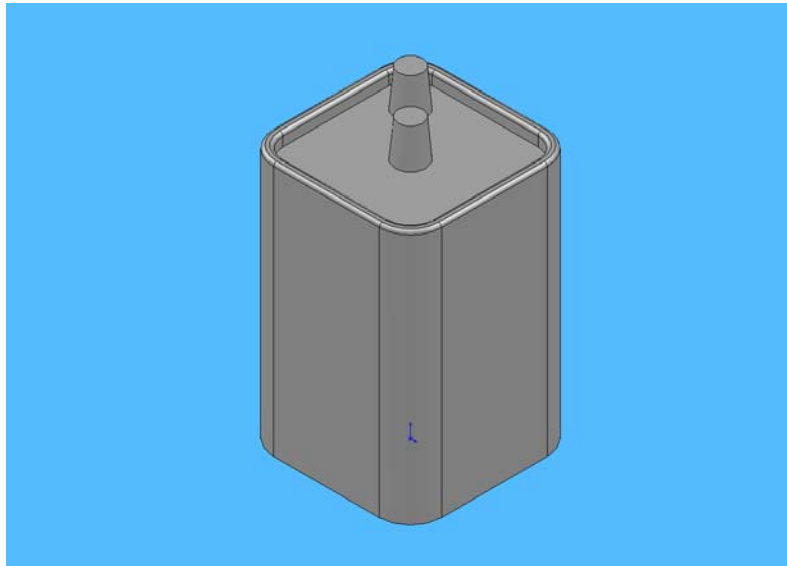
- MODELED THE CYLINDER AND CREATED A DRAWING FROM THE PART. MODIFIED THE DRAWING TEMPLATE TO INCLUDE PROPER UNITS (I.E. MM OR INCH DEPENDING ON THE APPLICATION) CREATED A COMPANY LOGO, LINKED FEATURES INTO THE TITLE BLOCK (I.E. COMPANY NAME, PART DESCRIPTION, PART NUMBER, DRAWN BY, DRAWN DATE AND MATERIAL). USED VIEW PALLETTE TO BRING IN VIEWS TO THE DRAWING TEMPLATE. ADDED THE CENTER LINES AND CENTER MARK TOOLS IN THE ANNOTATION TAB, TO THE .625 DIAMETER HOLE ON THE PART. LASTLY, ADDED A NOTE, AND LINKED THE MATERIAL PROPERTIES TO THE NOTE.

PROJECT 3/LESSON 21/EXERCISE 3.5: PRESSURE PLATE DRAWING



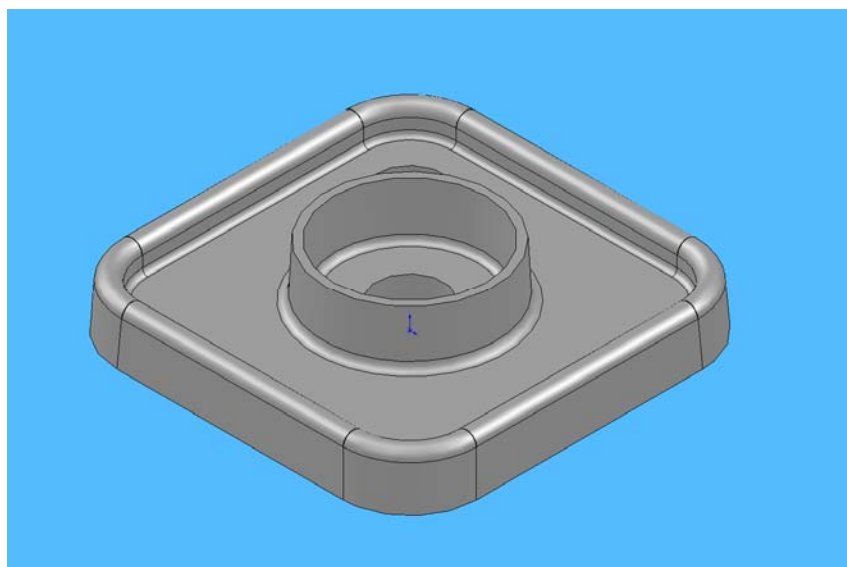
- MODELED THE PRESSURE PLATE AND CREATED A DRAWING FROM THE PART. MODIFIED THE DRAWING TEMPLATE TO INCLUDE PROPER UNITS (I.E. MM OR INCH DEPENDING ON THE APPLICATION) CREATED A COMPANY LOGO, LINKED FEATURES INTO THE TITLE BLOCK (I.E. COMPANY NAME, PART DESCRIPTION, PART NUMBER, DRAWN BY, DRAWN DATE AND MATERIAL). USED VIEW PALLETTE TO BRING IN VIEWS TO THE DRAWING TEMPLATE. ADDED THE CENTER LINES AND CENTER MARK TOOLS IN THE ANNOTATION TAB, TO THE .50 DIAMETER HOLES ON THE PART. MOVED DIMENSIONS FROM DIFFERENT VIEWS BY SELECTING THE VIEW, PRESS CONTROL AND DRAGGING THE DIMENSIONS TO ANOTHER VIEW. LASTLY, ADDED A NOTE, AND LINKED THE MATERIAL PROPERTIES TO THE NOTE.

PROJECT 4/LESSON 22/BATTERY



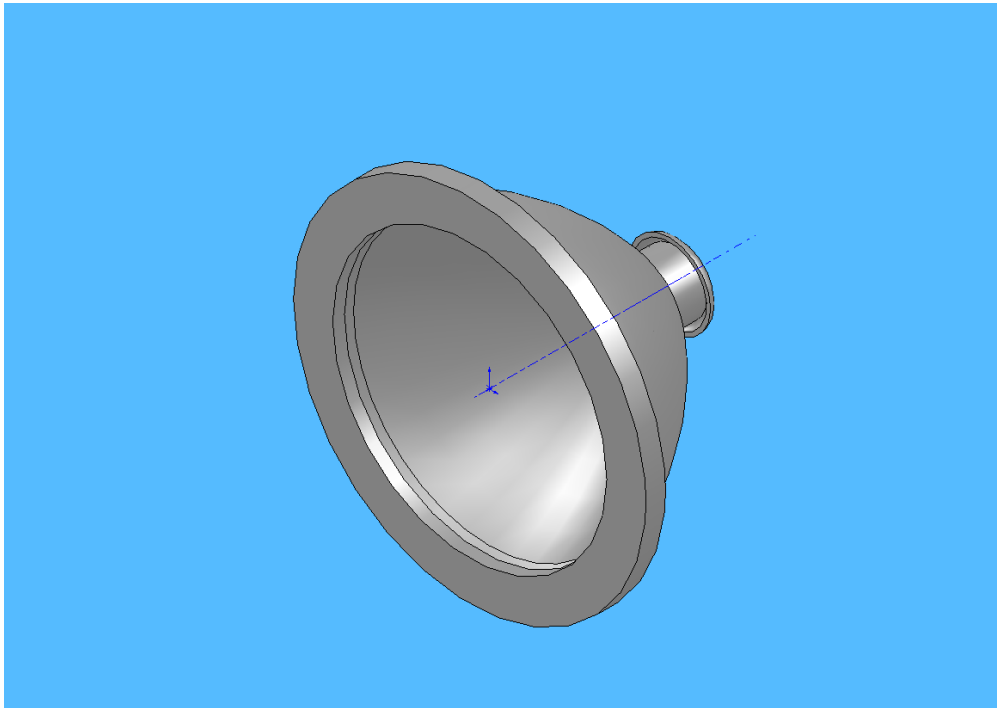
- USED SKETCH TOOLS (CONVERT ENTITIES, OFFSET ENTITIES) AND PARAMETRIC FEATURES I.E. 3-D EXTRUDE BOSS/BASE , EXTRUDED CUT, DRAFT, & EDGE BLEND COMMAND TO CREATE BATTERY. ADDIITIONALLY, LEARNED TO IMPLIMENT DESIGN INTENT IN MODEL, TO MAKE FOR EASE IN MODIFICATION OF THE BATTERY.

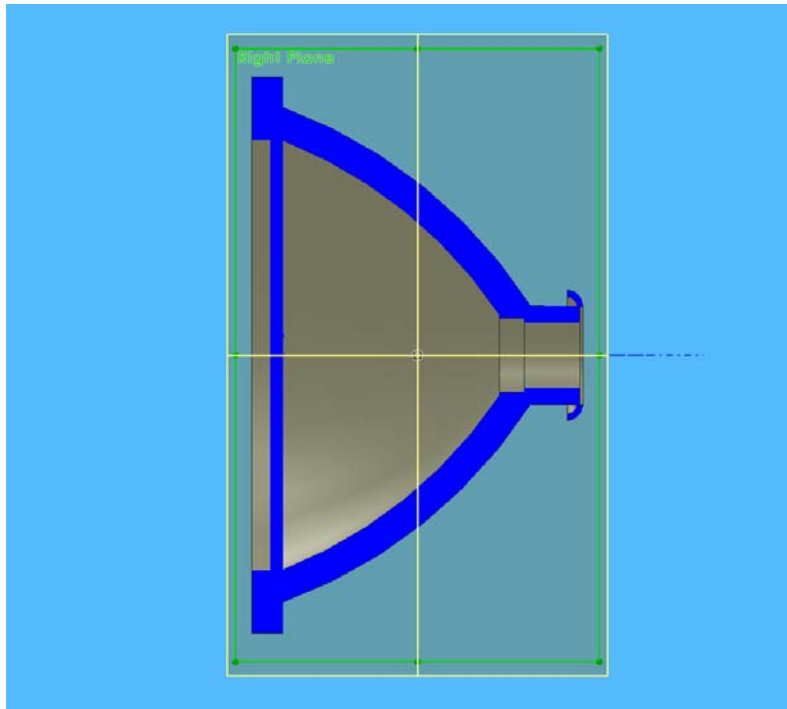
PROJECT 4/LESSON 23/BATTERY PLATE



- USED FEATURES FROM THE BATTERY PART, SAVED, AND MODIFIED TO CREATE THE BATTERY PLATE PART. SKETCH TOOLS (CONVERT ENTITIES, OFFSET ENTITIES, ADD RELATION TOOLS.) AND PARAMETRIC FEATURES I.E. 3-D EXTRUDE BOSS/BASE , EXTRUDED CUT, DRAFT, & FULL ROUND FILLETS, MULTIPLE RADIUS FILLETS COMMAND TO CREATE BATTERY PLATE. ADDIITIONALLY,

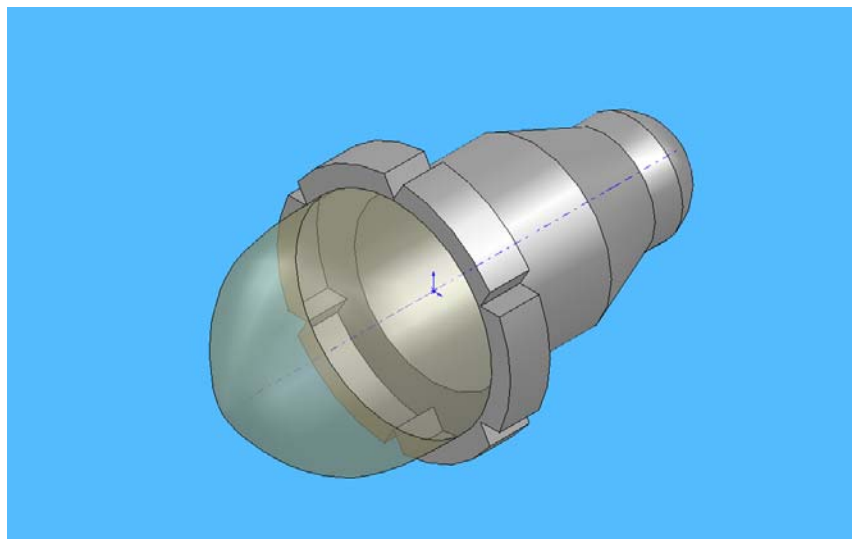
PROJECT 4/LESSON 24/LENS

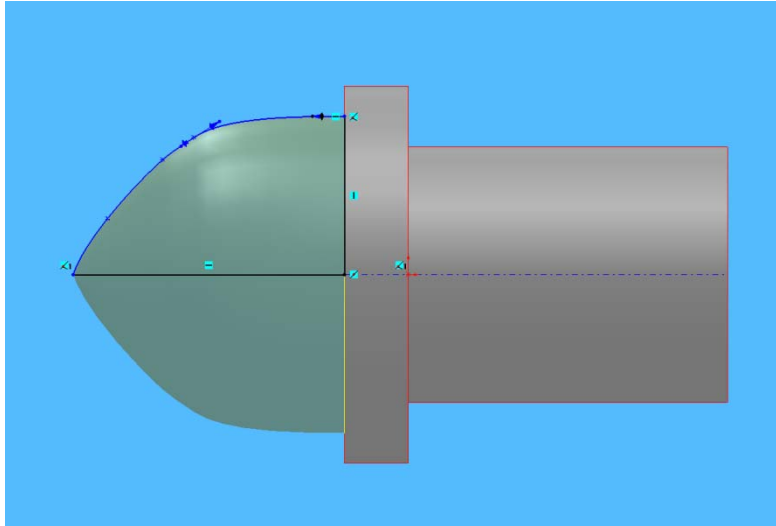




- USED THE SKETCH TOOLS (CONVERT ENTITIES, CONSTRUCTION GEOMETRY, OFFSET ENTITIES, OADD RELATION TOOLS.) AND PARAMETRIC FEATURES I.E. REVOLVE, EXTRUDED CUT, SHELL FEATURE, AND SECTION VIEW TO CREATE THE LENS.

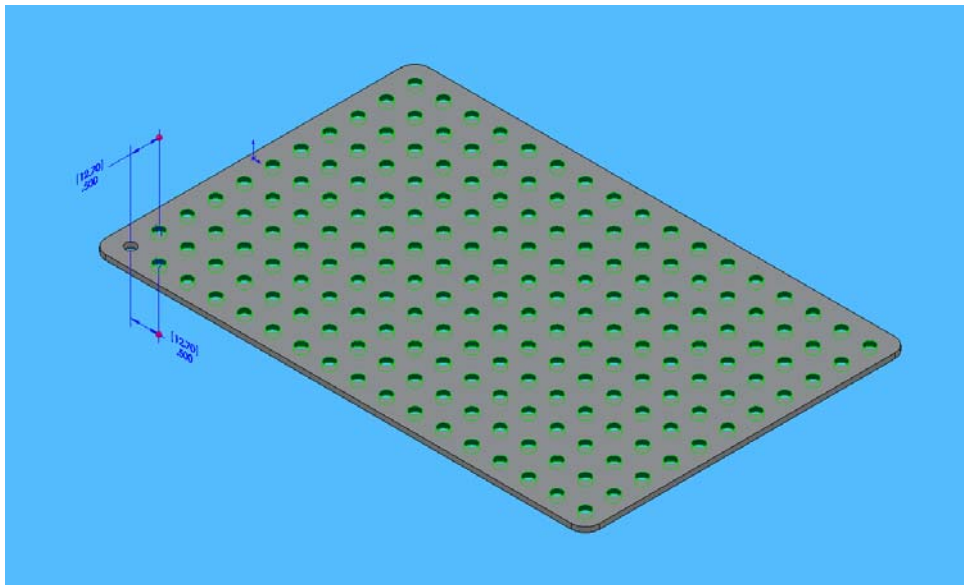
PROJECT 4/LESSON 25/BULB

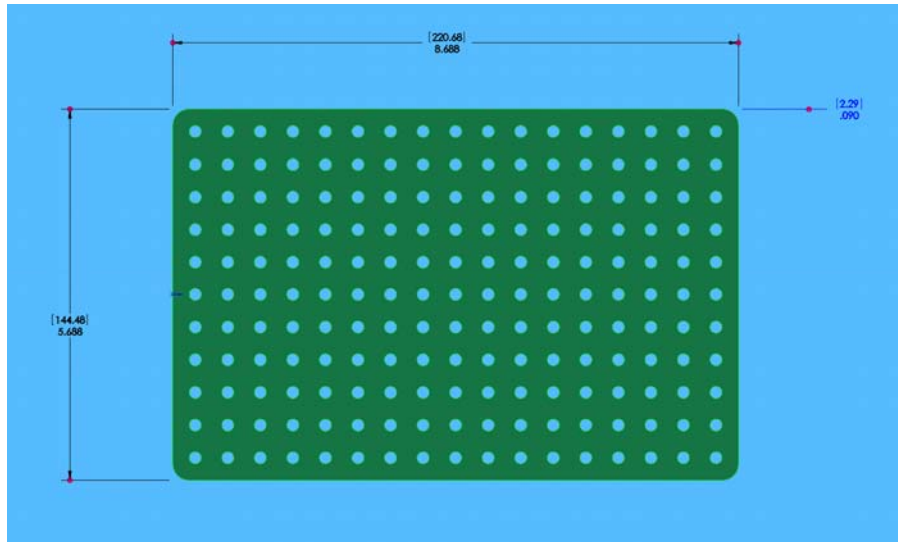




- USED THE SKETCH TOOLS (CONVERT ENTITIES, CONSTRUCTION GEOMETRY, OFFSET ENTITIES, ADD RELATION TOOLS, SPLINE TOOLS) AND PARAMETRIC FEATURES I.E. REVOLVE, REVOLVE CUT, EXTRUDED CUT, CIRCULAR PATTERN, DOME FEATURE AND APPEARANCE FEATURE TO CREATE THE BULB.

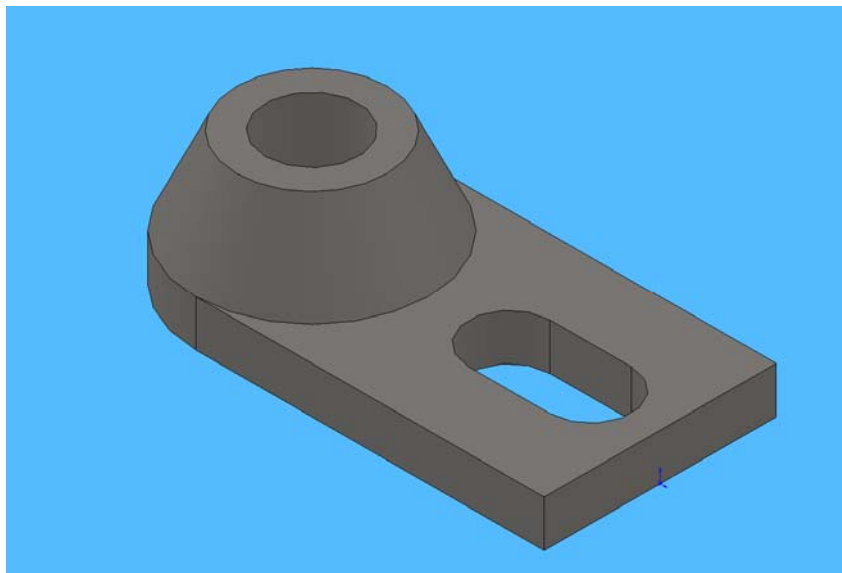
PROJECT 4/LESSON 26/EXERCISE 4.1: FLAT PLATE





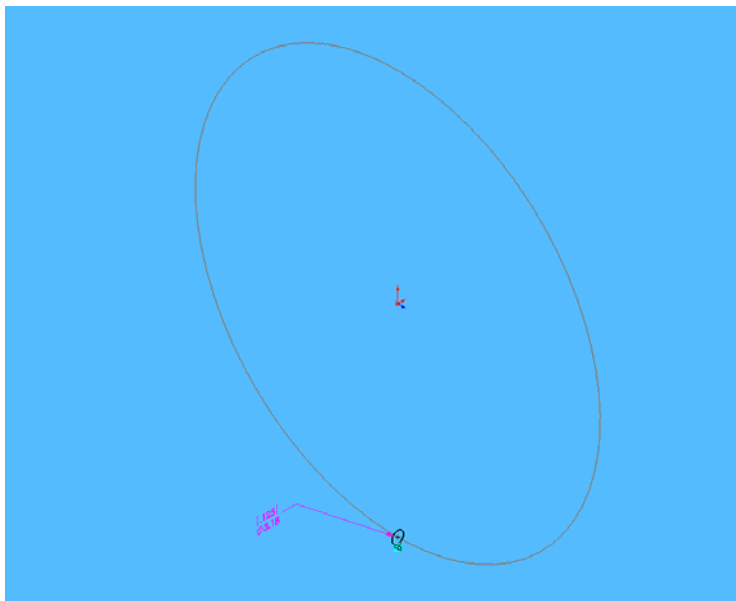
- USED SKETCH TOOLS, DUAL DIMENSIONS, AND PARAMETRIC FEATURES (I.E. EXTRUDE BOSS/BASE , EXTRUDED CUT, LINEAR PATTERN & FILLET COMMAND TO CREATE 1060 ALLOY FLAT PLATE.

PROJECT 4/LESSON 27/EXERCISE 4.9

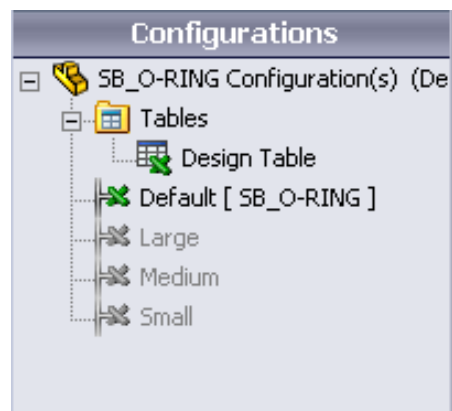


- USED SKETCH TOOLS, AND PARAMETRIC FEATURES (I.E. EXTRUDE BOSS/BASE , EXTRUDED CUT, REVOLVED TO PART.

PROJECT 5/LESSON 28/O-RING

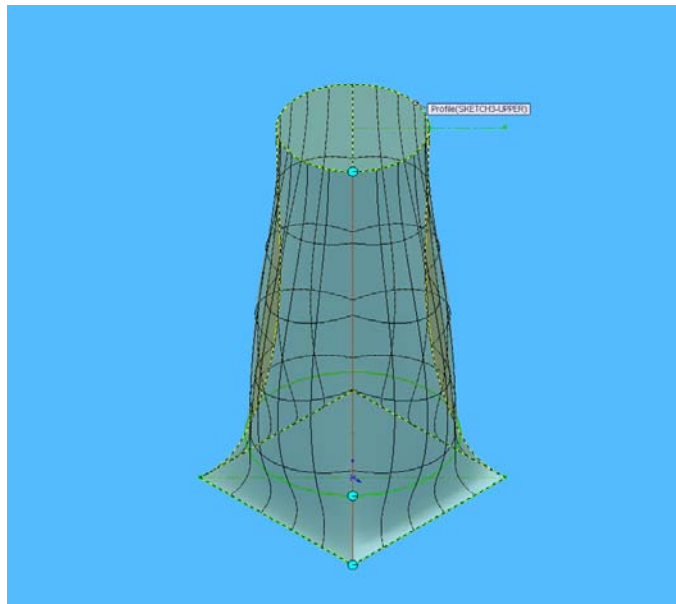
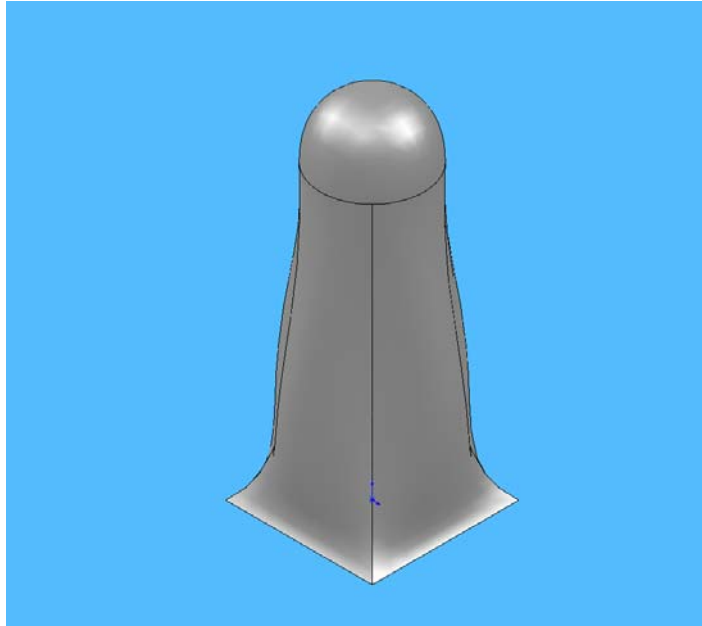


	A	B	C	D	E	F	G
1	Design Table for: SB_O-RING						
2		D1@SKETCH-PATH	D1@SKETCH-PROFILE				
3	Default	110.49	3.175				
4	Small	100	3				
5	Medium	150	4				
6	Large	200	10				
7							
8							
9							
10							



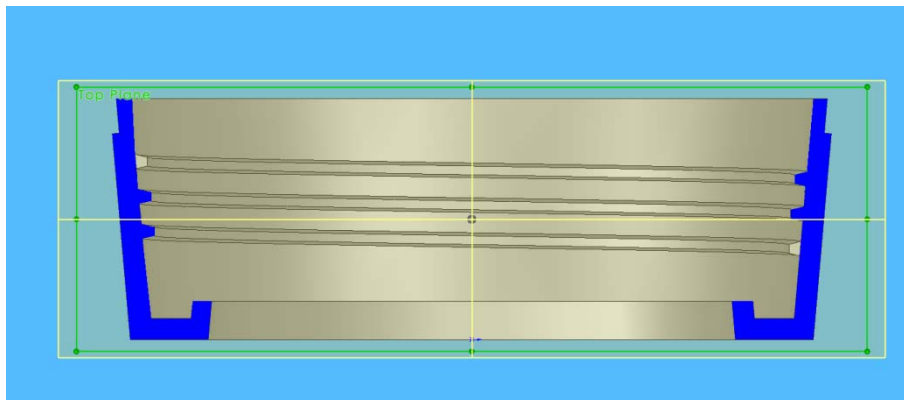
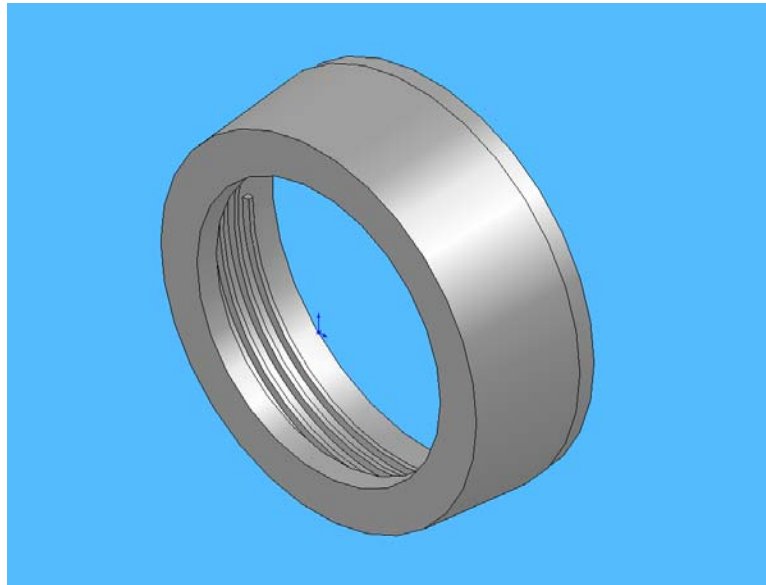
- USED SKETCH TOOLS TO CREATE TWO CIRCLES ON THE FRONT AND RIGHT PLANES. APPLIED A PEIRCE CONSTRAINT TO EACH SKETCH. USING THE SWEEP FEATURE, ALLOWED FOR THE CREATION OF THE O-RING. APPLYING A DESIGN TABLE, ALLOWED FOR THE MODIFICATION OF THE O-RING SIZE (I.E. SMALL, MEDIUM, AND LARGE).

PROJECT 5/LESSON 29/SWITCH



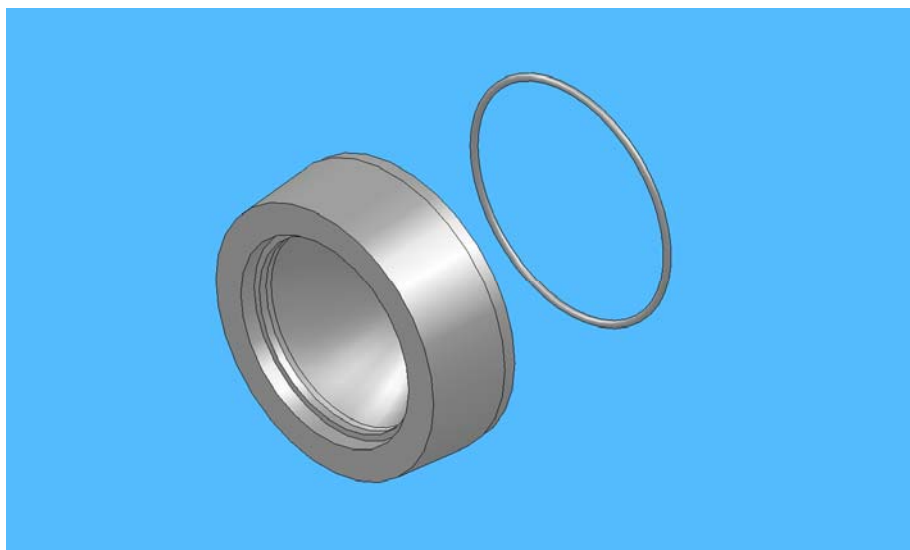
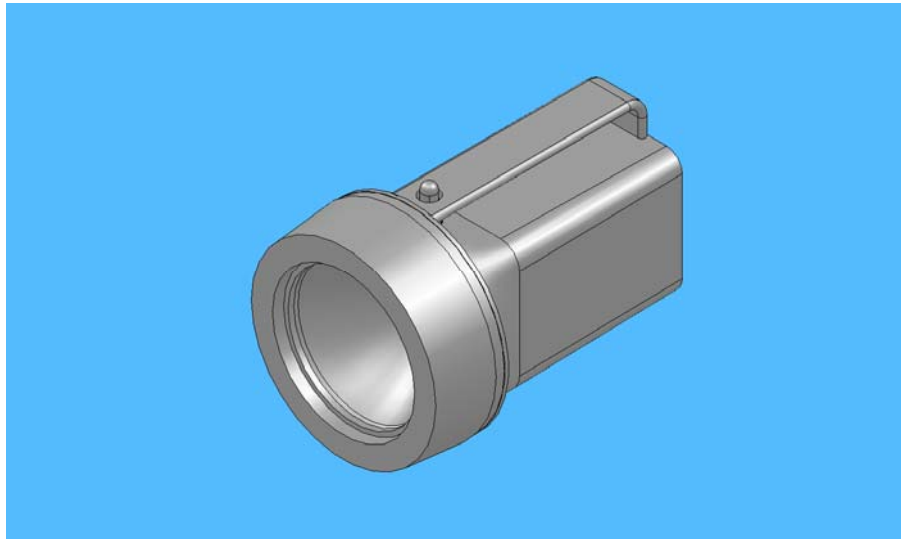
- CREATE TWO PLANES OFFSETED FROM THE TOP PLANE. USED SKETCH TOOLS (RECTANGLE, CIRCLES), AND PARAMETRIC FEATURES (I.E. LOFT AND DOME) TO CREATE SWITCH.

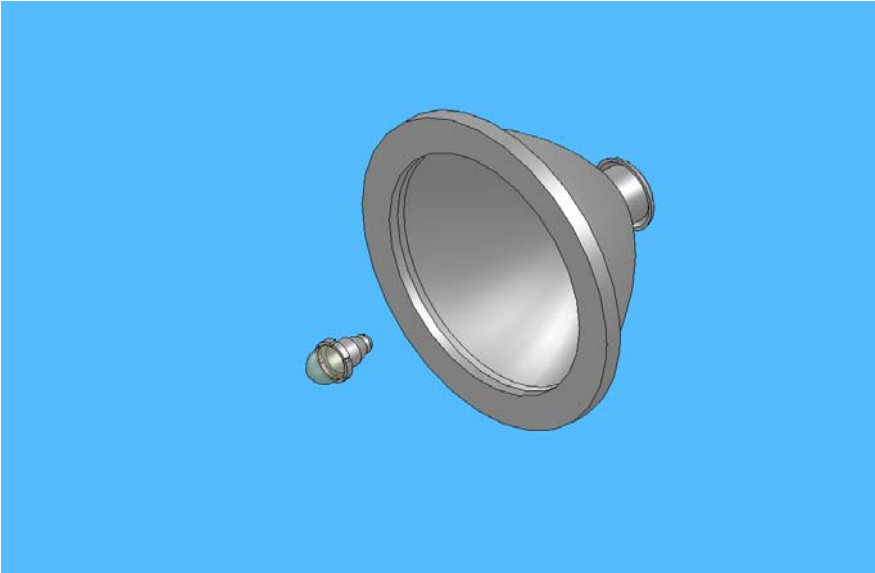
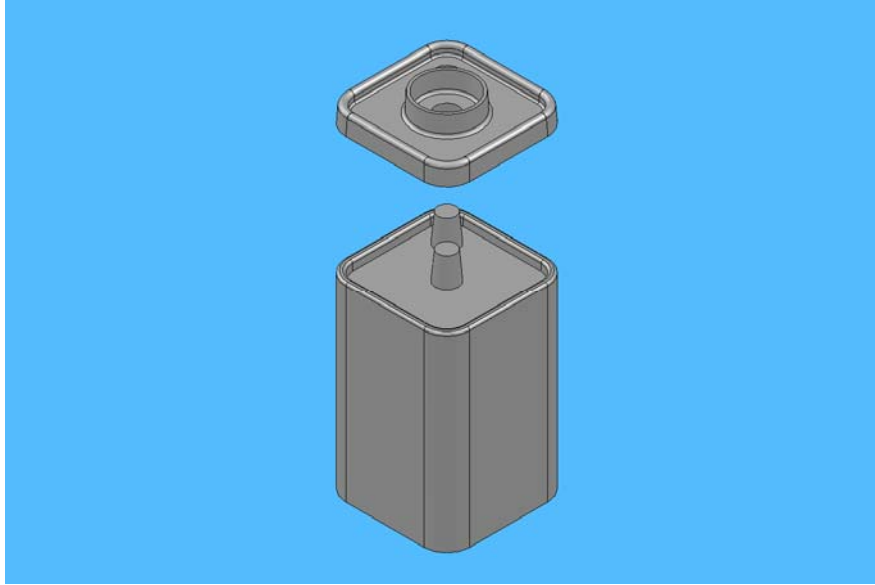
PROJECT 5/LESSON 30/LENS CAP

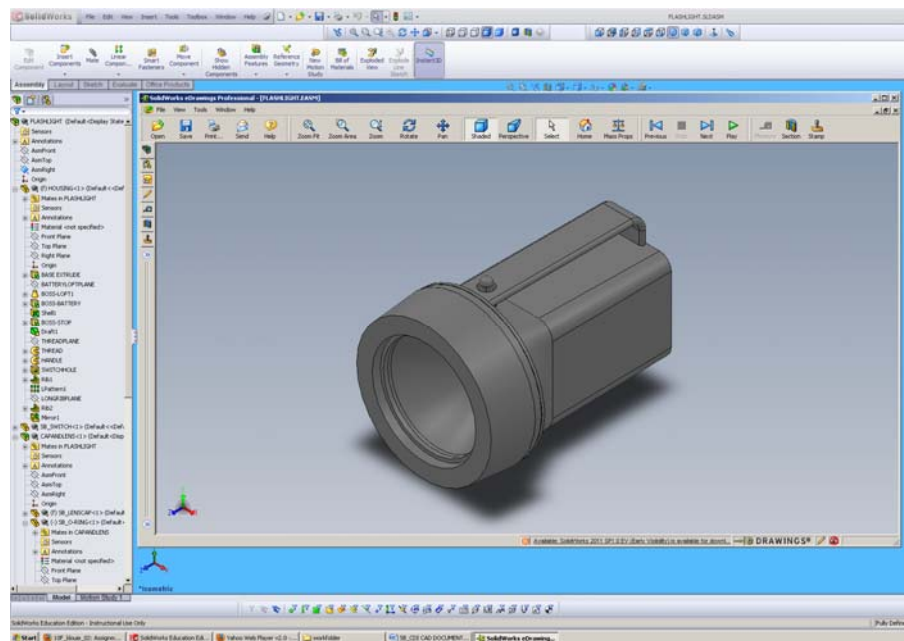
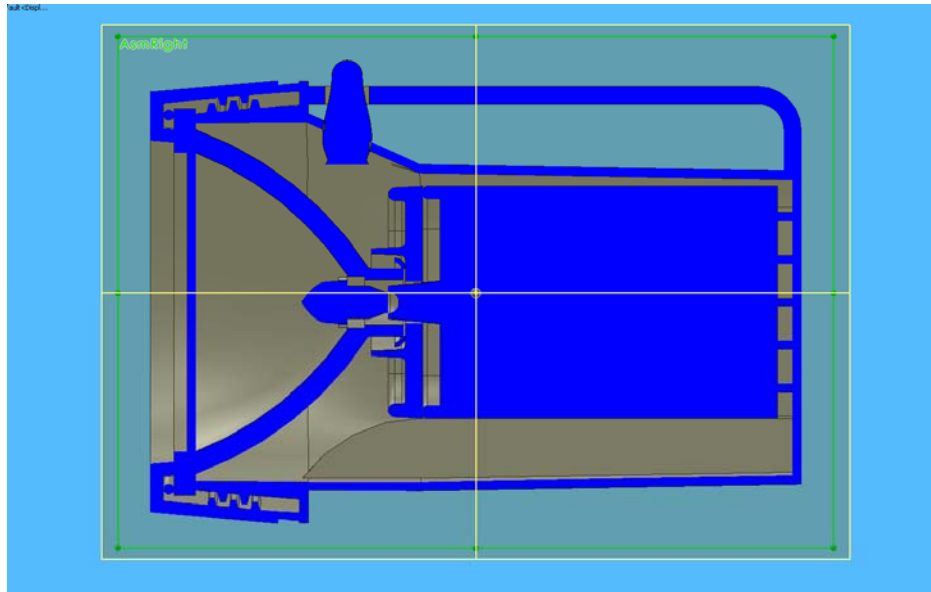


- USED SKETCH TOOLS AND PARAMETRIC FEATURES I.E. EXTRUDE BOSS/BASE , DRAFT, EXTRUDED CUT, SHELL, REVOLVE CUT, AND HELIX COIL CURVE TO CREATE LENS CAP.

PROJECT 5/LESSON 31/FLASHLIGHT ASSEMBLY

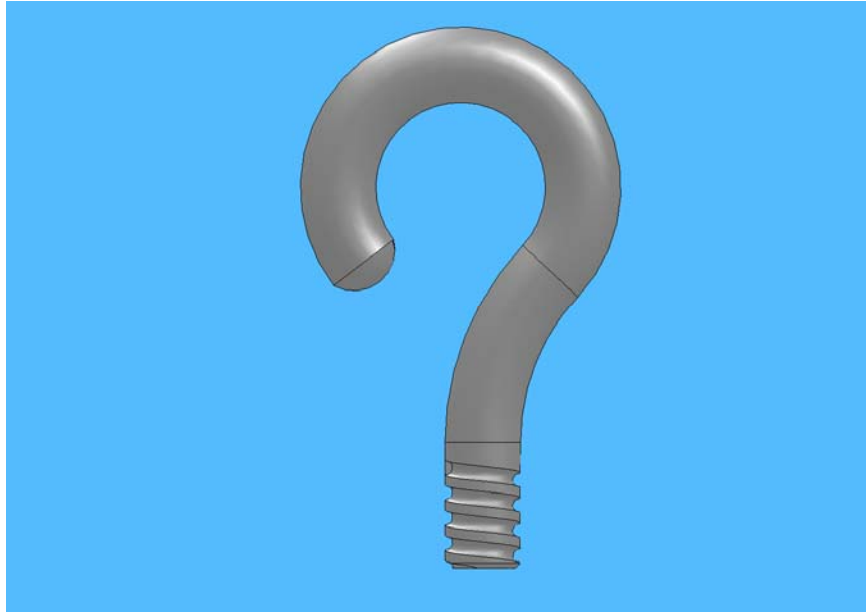






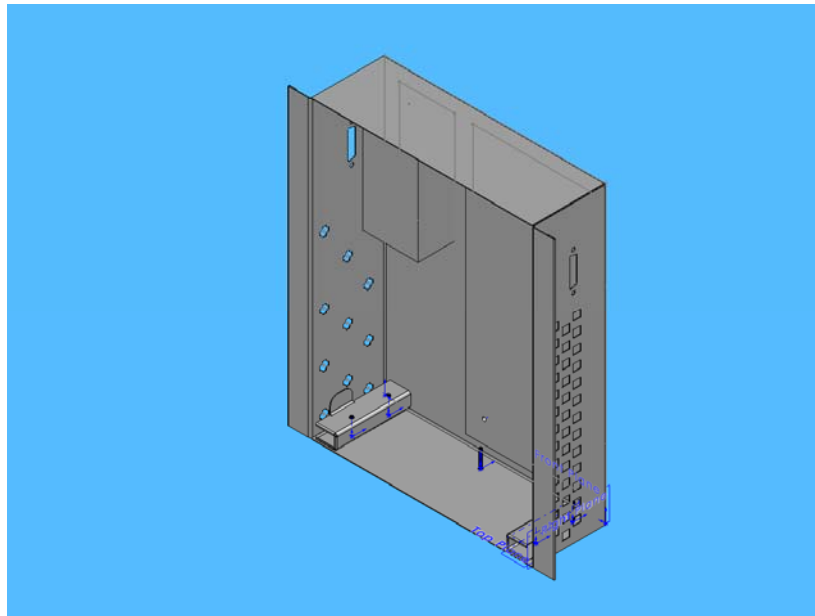
- CREATED SEVERAL SUBASSEMBLIES I.E. (LENS AND BULB, BATTERY AND PLATE, & CAP AND LENS) TO CREATE THE FLASHLIGHT ASSEMBLY. ADDITIONALLY, UTILIZED THE E-DRAWINGS TOOL TO SEND VITAL INFORMATION TO VENDORS REGARDING THE FLASHLIGHT HOUSING.

PROJECT 5/LESSON 32/EXERCISE 5.2: HOOK



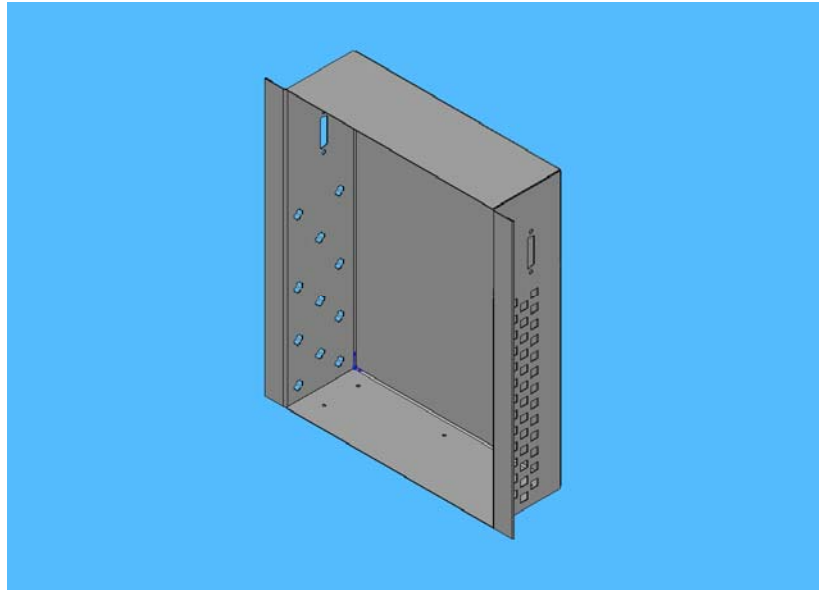
- USING VARIOUS SKETCH TOOLS (LINE, TANGENT ARC, CIRCLE) AND PARAMETRIC TOOLS (SWEEP, HELIX & SWEEPED CUT) CREATED THE HOOK COMPONENT.

PROJECT 6/LESSON 33/ELECTICAL BOX ASSEMBLY



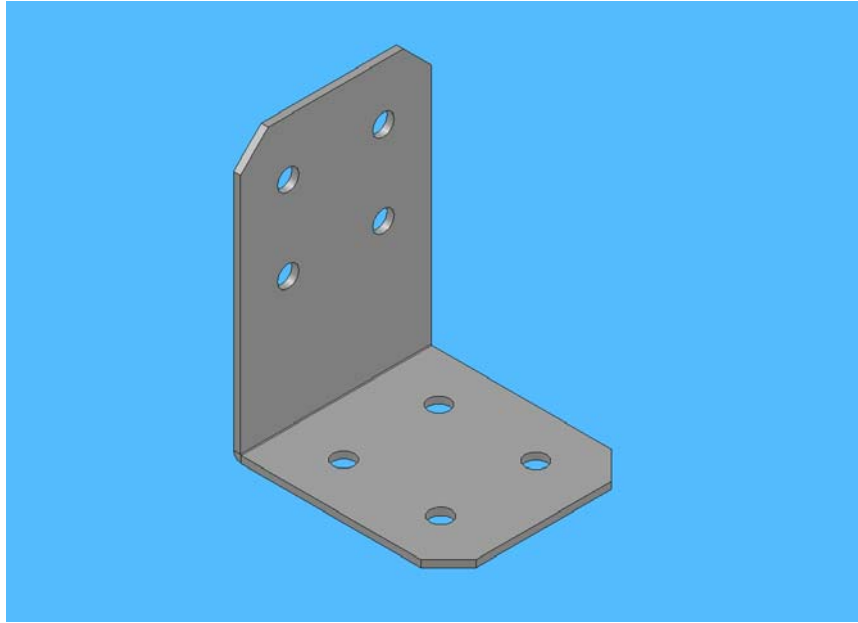
- USING VARIOUS TECHNIQUES SUCH AS TOP DESIGN, THE BOX, THE BRACKET, THE POWER SUPPLY, AND THE CONNECTOR WAS CREATED.

PROJECT 6/LESSON 34/BOX BASE (CABINET)



- USING VARIOUS TECHNIQUES SUCH AS TOP DOWN DESIGN, SHEETMETAL FEATURES, DESIGN LIBRARY FEATURES AND HARDWARE, THE CABINET WAS CREATED.

PROJECT 6/LESSON 35/BOX EXERCISE 6.5: L-BRACKET



- USING VARIOUS TECHNIQUES SUCH AS SHEETMETAL FEATURES, DESIGN TABLE, EQUATIONS, LINKS, AND DRAWINGS CREATED THE SHEETMETAL L-BRACKET.