Lab Assignment #8: Assemblies ME 170

Part I. Assemble our Electronic Product

1) Create a new folder

a) Start Fusion and open the Data Panel

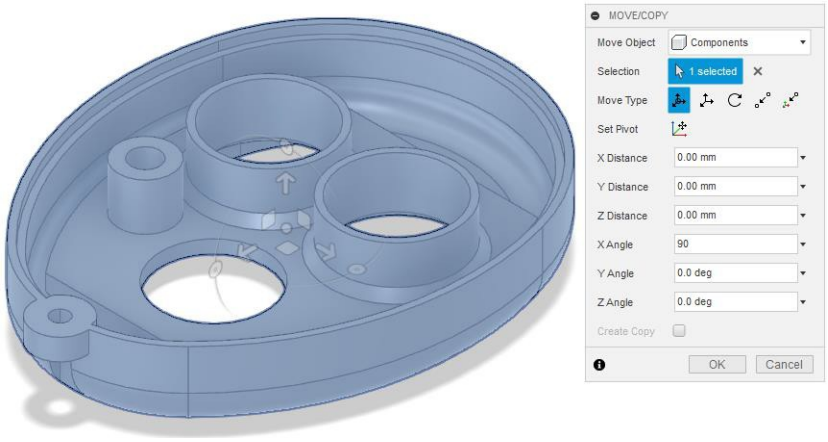
b) Navigate to your ME170 folder and create a new subfolder named “Lab 8”

c) Make a copy of all 9 Parts, setting the destination as the Lab 8 folder (make sure to use the most up to date version of each part) - Hold down Ctrl to select multiple files at the same time

d) Create a new file named “Electronic Product Assembly” and save it in Lab 8. Keep this file (and the Data Panel) open

2) Assemble the Outer Components

a) In the Data Panel, right click on Part 4 and select “Insert into Current Design.” Enter 90 deg as the X Angle and press OK

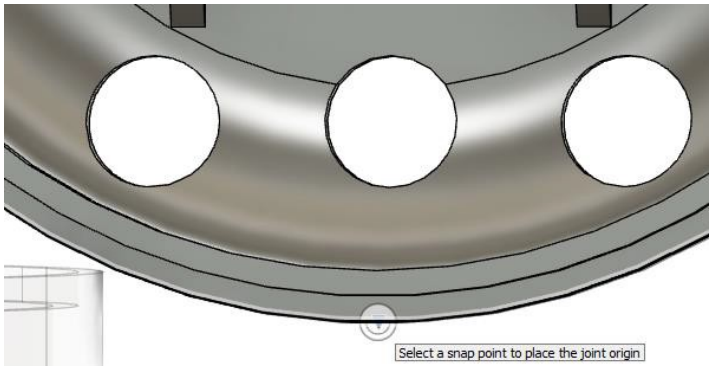


b) Right-click on the part name in the Browser and select Ground

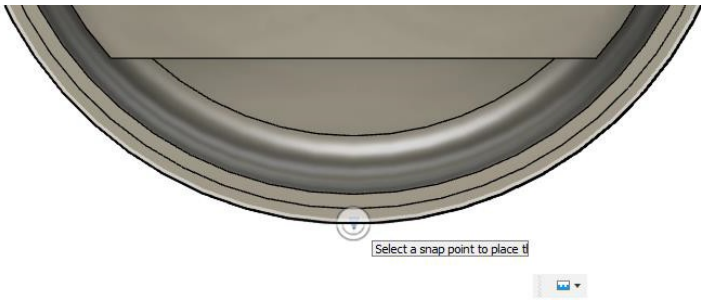
c) In the Data Panel, right-click on Part 5 and select “Insert into Current Design”. If needed, move Part 5 around so that your view is unimpeded by Part 4

d) From the Assemble section of the top toolbar, select the Joint tool  . This tool creates relationships between separate parts in an assembly, allows us to limit the degrees of freedom of the parts.

e) Select “Joint”. For Component 1 of the Joint, select the bottom point on the inside of Part 5



f) For Component 2, select the corresponding bottom point on the inside of Part 4



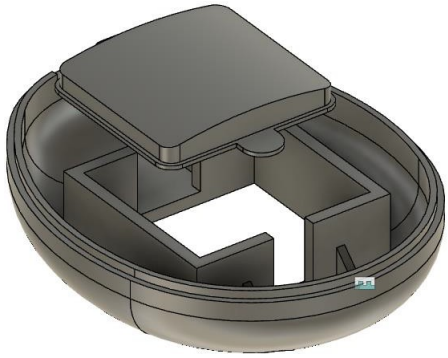
g) Enter 180 deg under “Angle” to make sure the parts line up, and press OK to create the Joint

h) Visually check that the two parts are assembled correctly (i.e. pan and rotate)

i) In the Browser, select the Eye next to Part 4 to Hide the part. This will allow us to fill the other parts without obstruction

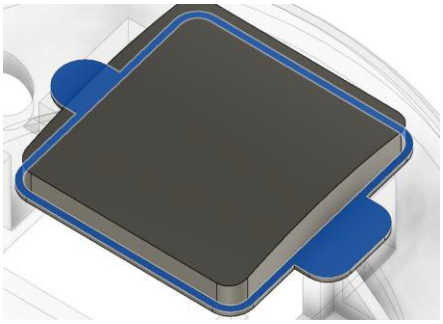
3) Assemble and Modify the Inner Parts

a) In the Data Panel, right-click on Part 6 and select “Insert into Current Design”. Set the Y Angle to 180 deg and press OK.

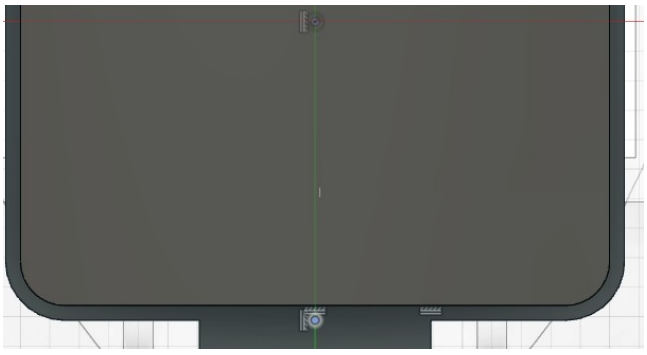


b) There is no convenient way to Join this part, so we will add one. In the Browser, right-click on Part 6 and select Edit in Place. The timeline at the bottom should change to Part 6’s timeline, instead of the Assembly file’s, and a black dot should appear on the Browser next to Part 6 to show it is Active (meaning editable)

c) Create a sketch on the following face:



d) Place a point and use the Horizontal/Vertical Constraint so that it is aligned with the origin and the top border of the inner rectangle

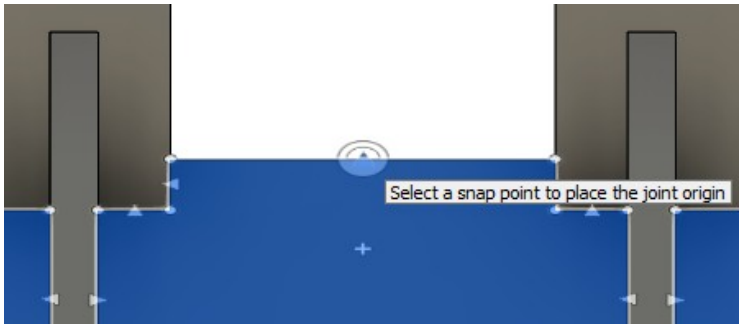


e) Press Finish Sketch and then right-click on Part 6 and select End Edit in Place

i) Note: if you Save the assembly and then open Part 6 independently, the new sketch will still be there since the file is linked

f) Select the Join tool, then choose the newly sketched point as Component 1

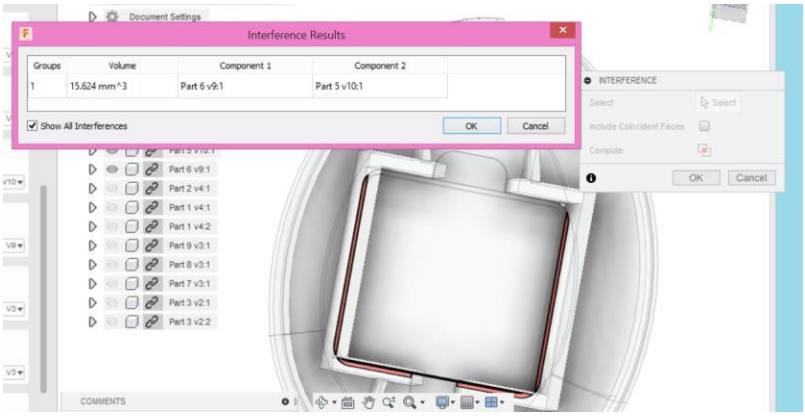
g) Select the corresponding inner-edge center point in Part 5 as Component 2



Tip: Select the face first, then the point is easier to select

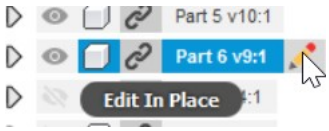
h) Make sure Flip is selected and enter 180 deg as the Angle, then press OK to create the Joint

i) This part would appear to be too large. To confirm this, perform an "Interference check". Click on INSPECT > Interference, on the main menu. Select the two parts and click on compute. Note the red highlighted area showing where the components interfere.



j) To fix this, 'Edit in Place' Part 6

k) Edit Sketch 1 by double clicking it in the timeline. Select 'Slice' and change the 21.5mm dimension to 20 mm. click finish sketch



l) Edit Extrude 1 and change the 10mm extrude to 9.625

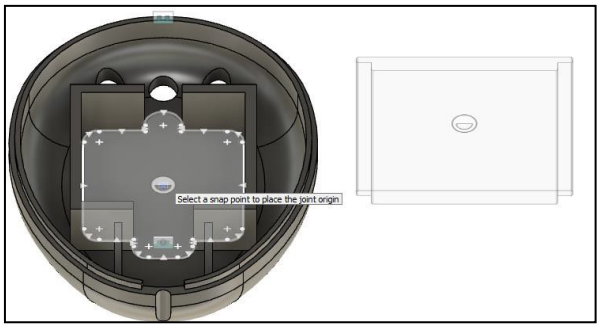
m) End Edit in place by clicking on the green check mark at the top of the window

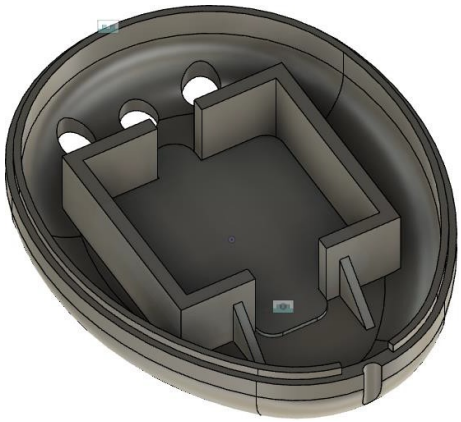
n) Perform another Interference check to make sure that you have fixed the problem.

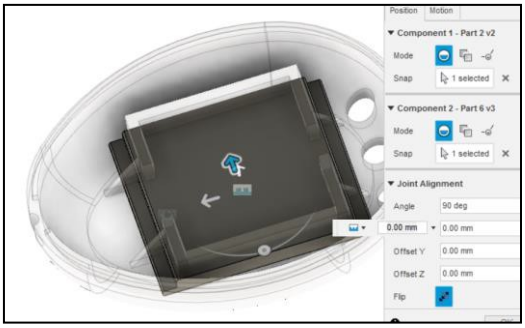
4) Assemble Part 2

a) Insert Part 2 into the Current Design, moving it as necessary to not block your view of the existing parts

b) Create a Joint where Component 1 is the center point on the bottom face of Part 2 and Component 2 is the center point of the inner rectangle of Part 6







c) Enter 90 deg as the Angle, flip if necessary to achieve the location shown in picture above, then press OK to create the Joint.

d) This part is clearly too large. To fix this, Edit in Place Part 2

Note on editing in assemblies: the timeline, at the bottom of the screen, gives you each joint as you create it. You can click on these and right click to edit them

e) Open the first Sketch of Part 2 and change the 28.8 mm dimension to 20.25 mm.

f) Change the first Extrude Distance from 12.5 mm to 10.75 mm.

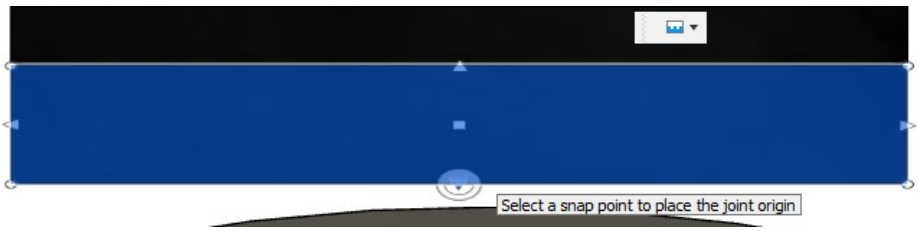
g) End Edit in Place, highlight all parts, and from the Inspect dropdown select Interference. Press Compute, and you should see that those changes removed any overlap as there no interferences.

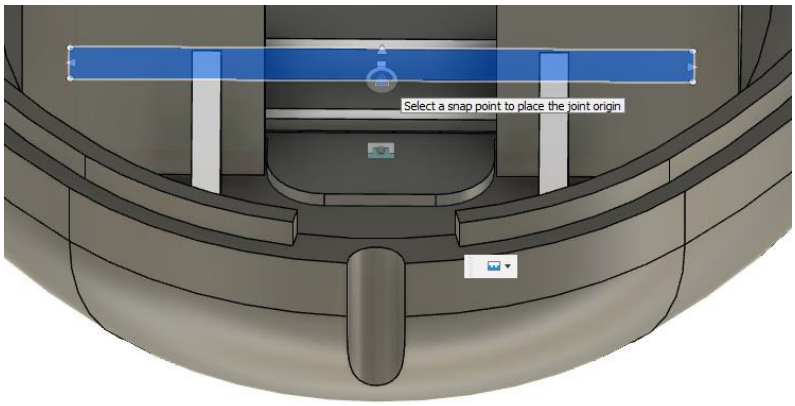
5) Continue Assembly Process

a) Insert two copies of Part 1 into the Current Design

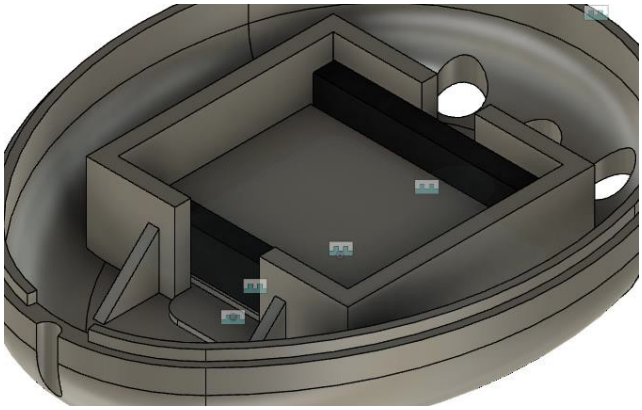
b) Create a Joint for each where Component 1 is the center of the bottom edge of Part

and Component 2 is the center of the side face of Part 2

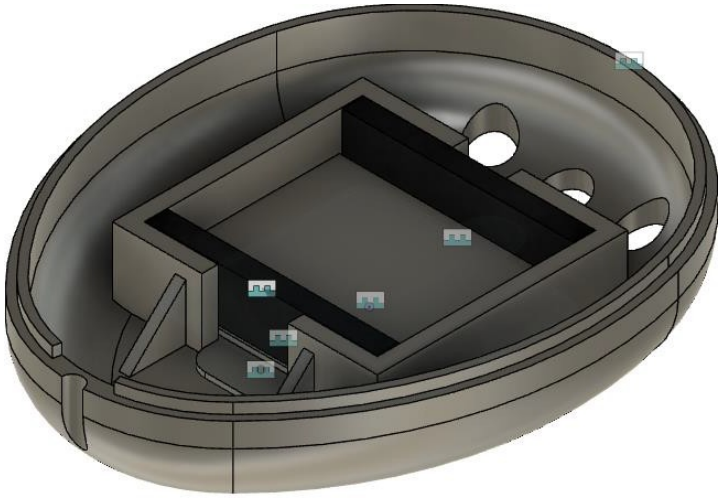




c) Flip as necessary and Press OK to create the Joint (make sure this is done for both copies)



d) Edit Part 1, and change the 4 mm dimension in the first sketch to 5 mm to give the part equal height. This should update both parts at once.

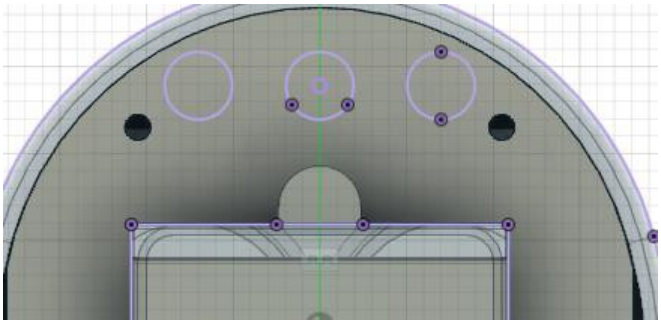


e) Insert Part 9 into the Current Design, moving it as necessary to not block your view

f) Create a Joint that matches the slots in Part 9 to the top profile in Part 1, ensuring the side with the flat side of Part 9 faces away from the holes in Part 5

g) Hide Part 5 and Unhide Part 4 and Edit Part 9. Create a Sketch on the flat, visible face

h) Unhide Part 5 and Project the three holes (select the Project Bodies option to make this easier), then Hide Part 5

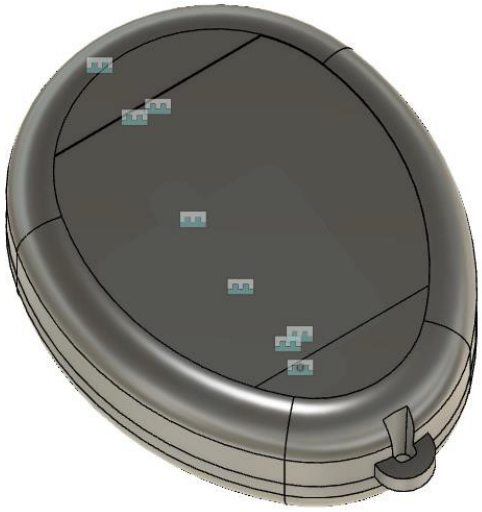


i) Insert Part 8 into the Current Design, moving it as necessary to not block your view

j) Create a Joint between the center of the bottom of the holes in Part 8 and the corresponding center of the circles projected onto Part 9

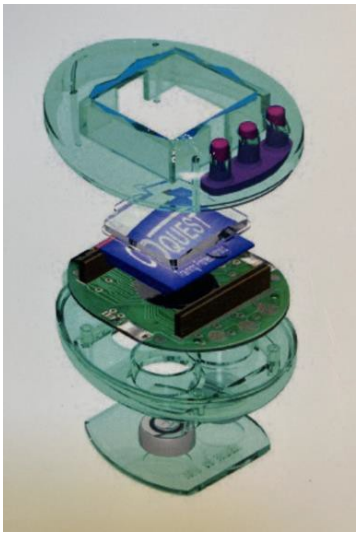
k) Unselect Flip and enter -90 deg as the Angle, then press OK to create the Joint

l) Insert Part 7 and use it to cover the bottom of Part 4



m) Now hide all but the last part and assemble the two batteries – Part 3

Part II Render the Model



Now we are going to add some "appearances" to the product and try to match those of the physical product in the photo on the right. Right click on each part in the Fusion "Browser" and select "Appearance". For each part, using the Appearance dialog box, follow this same procedure for the first part and color to suit:

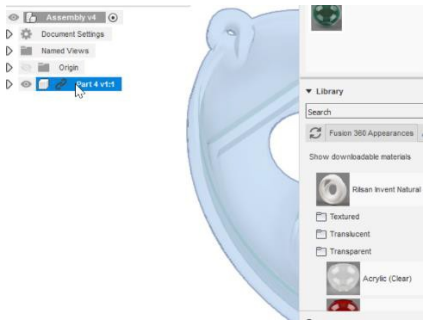
a. Go to the library and select Plastic -> Transparent -> Acrylic clear.

b. Drag the thumbnail image from the library and up into the "In This Design" window of the dialog box.

c. Right click to Duplicate then right click on the duplicate to Edit. Using the color selector choose a light green color and try to match what you see in the photo.

d. Drag the thumbnail onto the shaded CAD part model and it will recolor the part.

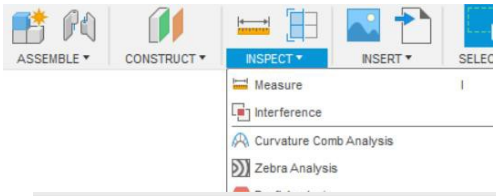
e. Repeat for other parts. You should end up with the assembled product looking something close to this:

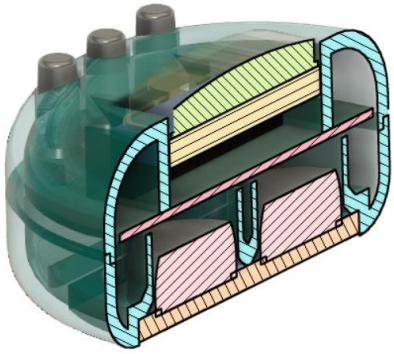


Part III. Create a Cross-Section through the CAD model

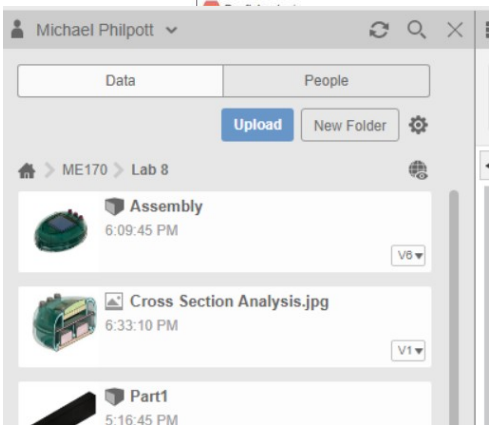
We will take a look inside the assembled product to make sure everything fits:

a. Click on INSPECT on the top menu and then choose "Section Analysis". Select a plane or face in the XZ direction and use the arrows to move the section through the middle of the batteries.





b. Note that the batteries are clearly too big. With the cross-section showing, "Edit in Place" the final feature, Hole 1, and change the diameter of the holes to match the battery, 12mm (was 10mm). Go to "End Edit in Place" by right clicking on Part 3.



c. Now add the screen image of the corrected cross-section view, for grading, into the Lab 8 folder:

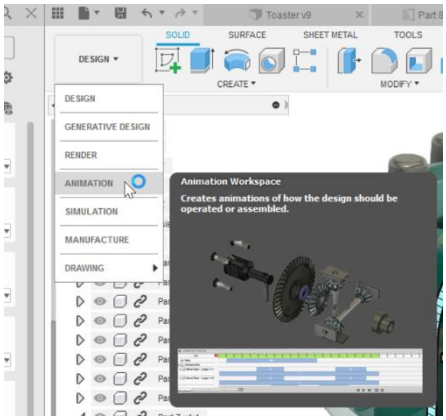
   i. Go to File -> Capture Image

   ii. Select "Save project to cloud" in the dialog box

   iii. Name the file "Cross Section Analysis"

   iv. Click ok, wait as it uploads, and check that your Lab 8 folder has the image.

Part IV. Create an Exploded-View Assembly Drawing



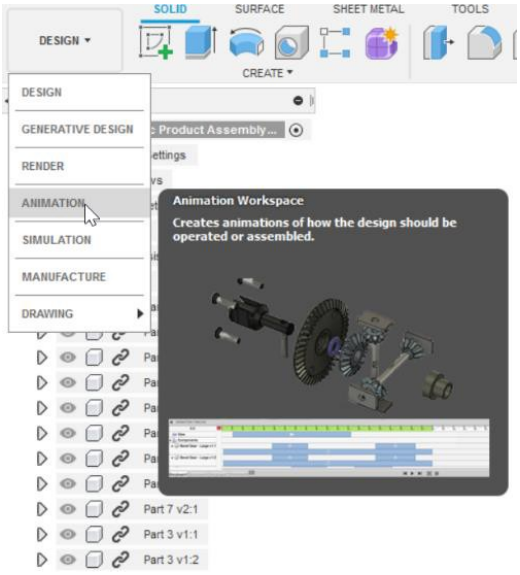
Engineering "drawings" are conventionally black & white line drawings without color or shading. **Part Drawings** require hidden lines to be shown. **Assembly Drawings** may optionally show hidden lines but generally it's clearer not to. So, "wireframe with visible edges only" is the default in the UofI Assembly Template that we will use. Notice there is no Units and Tolerance section in the titleblock as Assembly Drawings do not include dimensions or tolerances.

Two forms of Assembly Drawing are generally used as aids to assemble a product once all the parts have been fabricated:

- An Exploded View Assembly Drawing - with Bill of Material (or parts list)
- A Cross Section Assembly Drawing – showing details of adjacent parts as assembled.

Both these assembly drawings are shown in the Appendix – Master Drawings. As with the previous lab, your goal here is to replicate these master drawings.

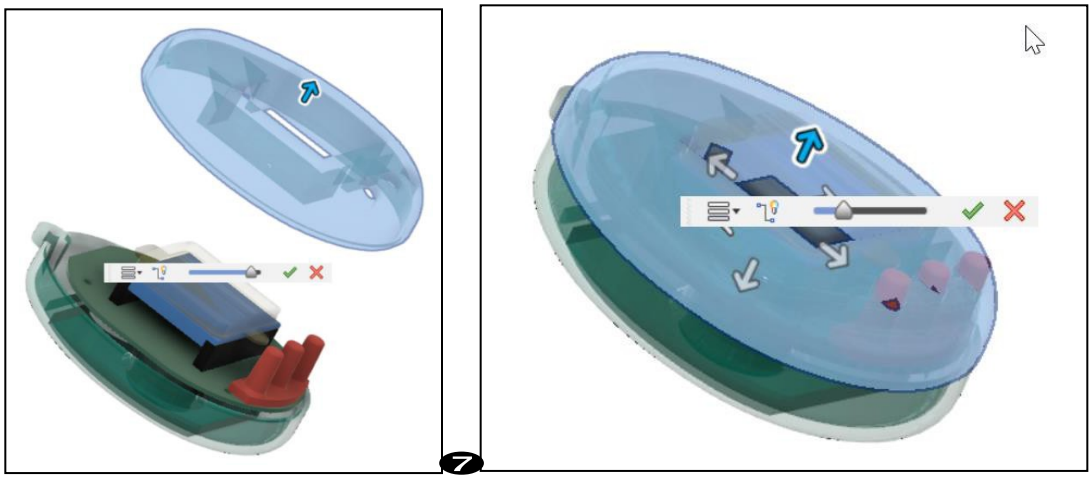
First download the "UofI Assembly Template" from the assignment link in Compass, and upload it to your Lab 8 folder.



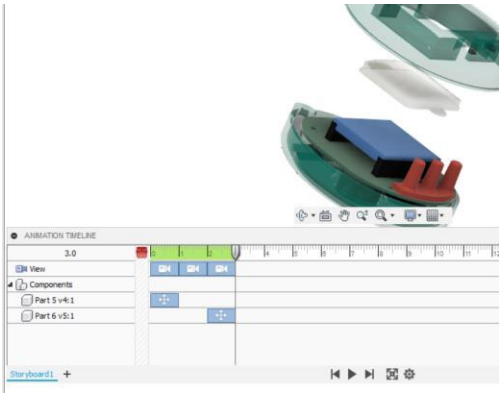
a) In the top left of the screen, change "Design" to "Animation" (the Animation space is used in Fusion 360 to create exploded views and associated assembly drawings).

b) Click on TRANSFORM on the top menu, and use the Manual Explode tool to move the components of the Assembly one-by-one to create an exploded view.

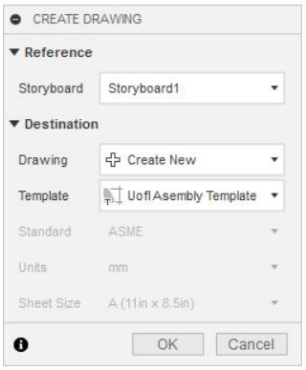
c) Select the vertical arrow each time then move the slider to the right until the part is clear of the previous part, then click the green check mark. Copy the Exploded View assembly in the Appendix.

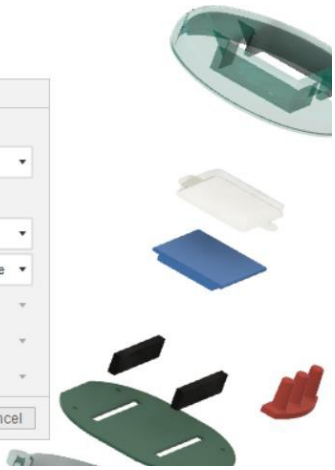


d) Note that in the “Animation Timeline”, at the bottom of the screen, you create an event of 1 unit of time each time you manually add a component. You can drag can play it back by clicking the movie playback arrow. You can also save it as a movie.



You may want to use rendered explode animations in your Final Presentations. However, our goal here is to create an Assembly Drawing as official documentation for the product. An exploded view assembly should aim to separate the parts out as much as possible with little to no over-lap of parts. Once all parts have been moved, select File > New Drawing > From Animation

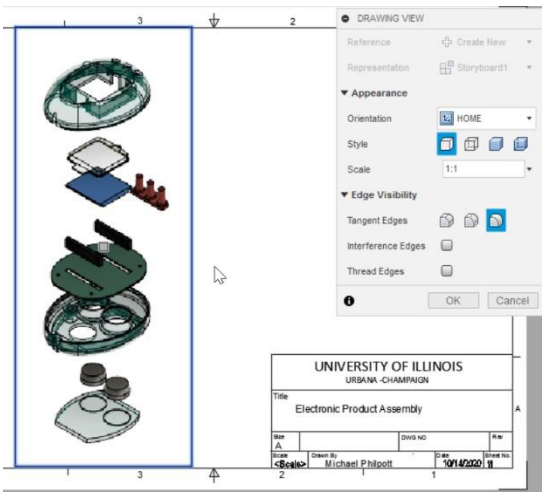




e) The drawing file will use the exploded view from the Storyboard you select in the Create Drawing dialog box (Storyboard1 by default). Select the “Uof I Assembly Template” as the template and click ok. Give it a minute for the drawing pop up.

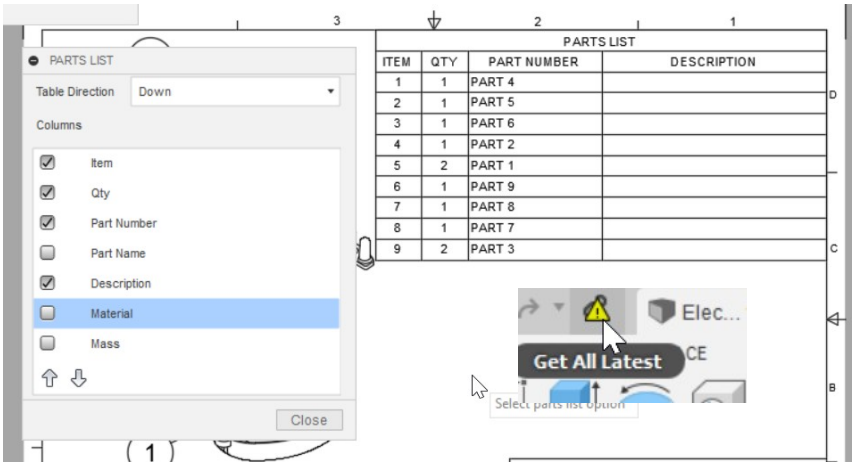
f) Using the Drawing View dialog box, select the scale to 1:1 and click ok. If necessary select a different orientation to match the view shown in the Exploded view in the Appendix. Click ok. You can move view as required after you click ok.

g) At this point it’s probably a good idea to save and name the file “Electronic Product Assembly Drawing”.

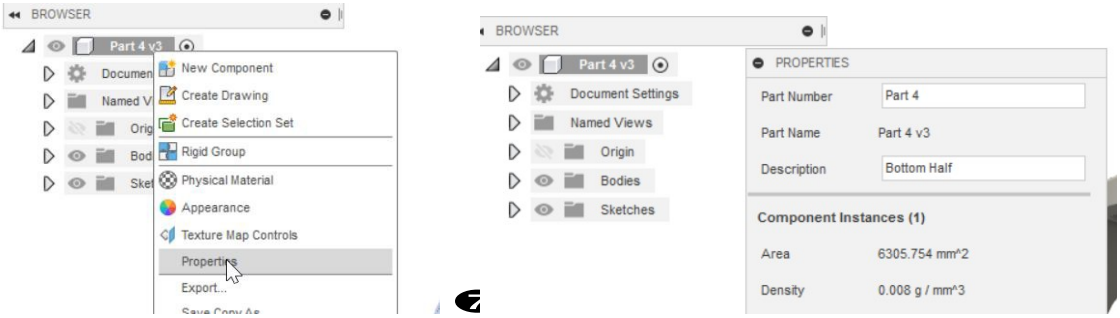


h) Click on TABLES > Table up on the main menu bar and click on the top right corner of the drawing to attach the BOM (Parts List)

i) Edit the table (double click) to include only those items check marked in the figure (i.e. item, qty, part number, and description). Clean up the table by resizing the columns (single click and drag handles) allowing space to the left for the exploded view. Again, match the Appendix.



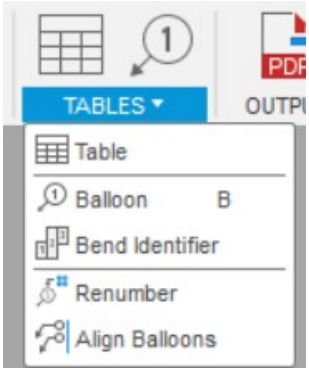
j) Add the part "Descriptions" to the Description field in the table by add the Description property to each of the parts. To do this, Open each part in the Design space, right click on the Part name in the Browser and select Properties. Fill in the descriptions from the Appendix for each Part 1 through 9. You will have to save each part and then click on "Get All Latest" to update the referencing for both the assembly model and the assembly drawing. When this has been done, the Parts list should update to include the part descriptions.



k) Balloons should have automatically appeared to identify the parts. ASME standards require balloons to be aligned without leader lines crossing over each other. You can use "Align Balloons to help with this and/or manually select and drag balloons to clean up and match the Appendix.

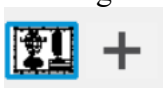
l) Finally change the title of the drawing in the title block to "Electronic Product Exploded View Assembly" as in the Appendix and check that everything else matches.

m) Save the drawing.



Part V. Create a Cross Section Assembly Drawing

a) We are going to add this to the existing Assembly drawing as an additional sheet (Sheet 2)

b) Click on the large Plus sign on the bottom of the exploded view assembly drawing just created.  A new empty sheet will be created. Edit the Title to "Electronic Product Cross Section Assembly".

c) Right click in the middle of the empty drawing and select Drawing Views > Base View. Select "UofI Assembly Template", click OK. Give it a minute to update.

d) Place the views with the parent TOP as the front view scaled 1:1 and two section views (using "Drawing Views" > "Section View" as in previous part drawing lab). Match the Cross Section assembly drawing in the Appendix i.e. front view in plain view, right view as section view, and bottom view as section view.

e) Edit the Cross Section titles to standard font height of 0.12 if necessary (the default appears sometimes to be 0.24 even though settings are 0.12…a bug I think!)

f) Add some balloons to identify key parts in each cross section by manually clicking on Tables > Balloons and placing as shown in the appendix. Balloons are not a requirement for Cross Section assembly drawings but they are helpful in identifying key parts and clarifying their assembly location/mating characteristics.

g) Save a PDF file of your two sheet Assembly Drawing (i.e. Output > Output PDF from the tool bar, top right) to your local computer.

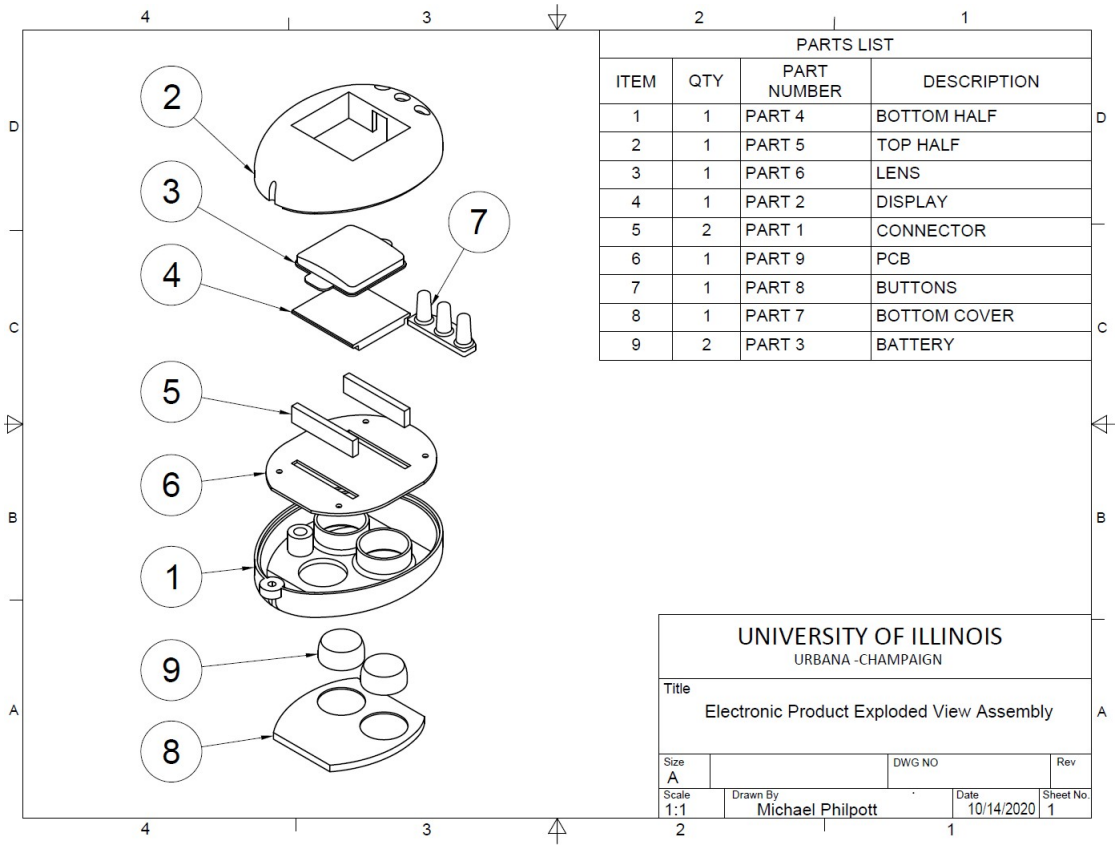
h) Save the drawing.

Part VI Submission Requirements

a. Make sure all your files are saved in the Lab 8 folder and that the folder is shared with your TA.

b. Go to ME170 Blackboard website and the CAD LAB Assignments content area. Click directly on the "CAD LAB 8" assignment title and use the "Browse My Computer" button to upload your pdf file version of your drawings. As before, please do not go back and change your files in the Fusion Lab folder (Lab 8).

Appendix – Master Drawings



PARTS LIST			
ITEM	QTY	PART NUMBER	DESCRIPTION
1	1	PART 4	BOTTOM HALF
2	1	PART 5	TOP HALF
3	1	PART 6	LENS
4	1	PART 2	DISPLAY
5	2	PART 1	CONNECTOR
6	1	PART 9	PCB
7	1	PART 8	BUTTONS
8	1	PART 7	BOTTOM COVER
9	2	PART 3	BATTERY

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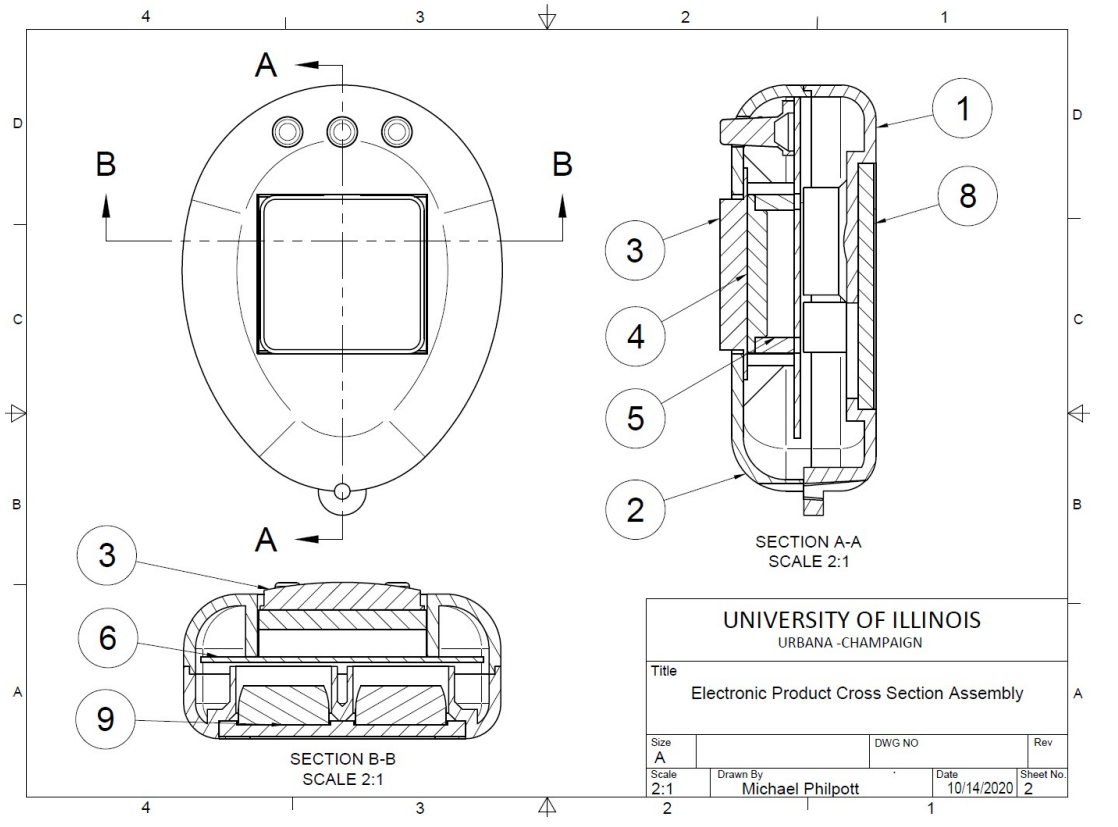
URBANA -CHAMPAIGN

Title: Electronic Product Exploded View Assembly

Size		DWG NO	Rev
A			

Scale	Drawn By	Date	Sheet No.
1:1	Michael Philpott	10/14/2020	1



SECTION B-B
SCALE 2:1

SECTION A-A
SCALE 2:1

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Size		DWG NO	Rev
A			

Scale	Drawn By	Date	Sheet No.
2:1	Michael Philpott	10/14/2020	2