

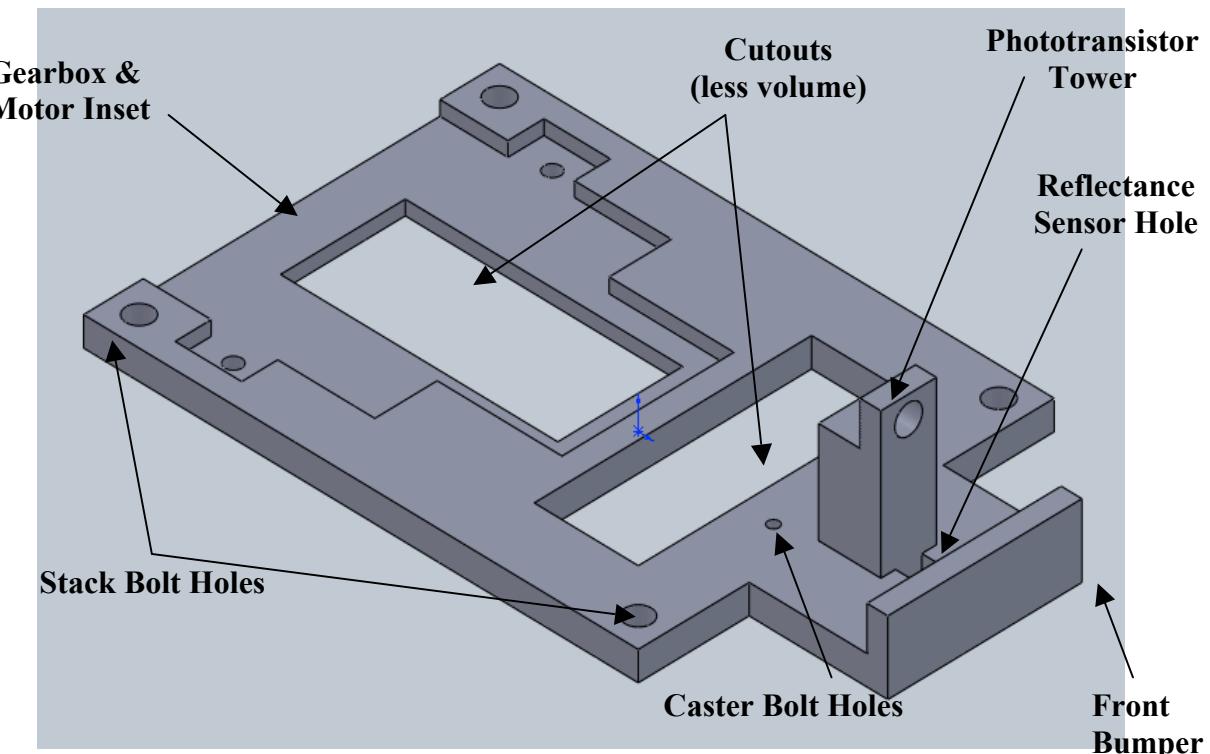
E11: Autonomous Vehicles

Fall 2010

Harris & Lape with Keeter & Ong

Lab 2: 3D CAD and Printing

The goal of this lab is to create a robot chassis in SolidWorks that can be printed on HMC's 3D printer. When you are done, the chassis should look like the one below.



Introduction

SolidWorks is a computer-aided design (CAD) program for modeling 3-dimensional objects. Using CAD tools, engineers can create models ranging from a single component to a fully-assembled machine. In this lab, you will create a single part: the chassis of your robot. When designing a part in SolidWorks, you will start with a two-dimensional drawing, called a *Sketch*. This sketch is then converted into a *Feature* to give it depth in the third dimension. By extruding sketches on various planes, complex parts can be created.

Constraints

CAD tools are meant to create parts for the real world, which means that each part must be fully dimensioned. Under the hood, SolidWorks stores a set of constraints that define a drawing. These constraints specify length, orientation, angle with respect to other objects, or many other characteristics. It is important to make each drawing fully defined, so that there is no uncertainty in the shape of the part. A fully defined line will show up in black; an undefined line will be drawn in blue.

3D Printing

A 3D printer is an additive manufacturing technology that builds up a 3-dimensional object with successive layers of material. The HMC Dimension SST 1200 3D printer constructs objects from white ABS plastic. The printer also uses a soluble black support material, which is used to hold up the building material. This support material is then dissolved away. Since a part is built upward in 10-mil layers, any layer that is larger than the lower layer requires support material to hold itself up while printing. The process costs approximately \$10/in³ of building material. Support material is less expensive, but it is nevertheless a good idea use no more than necessary.

To minimize printing time and cost, you will build a chassis that is made as thin as is practical and has cutouts to reduce volume. The design consumes less than 2 in³ of printing material, and prints in about 2 hours.

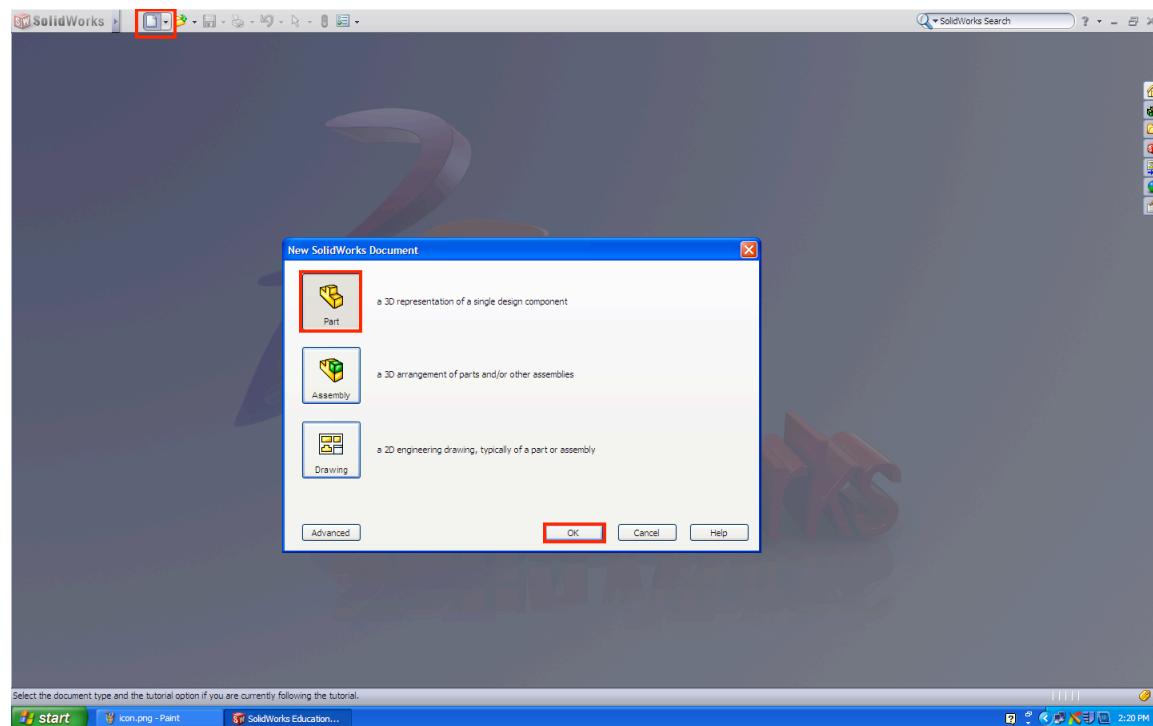
Getting started

First, log in to a computer in the Engineering Computational Facility (ECF). Double-click on the SolidWorks icon:



This will open SolidWorks Education Edition 2009-2010. [*** update](#)

The entire chassis will be a single part. Click on the New Document button in the upper left corner. Make sure that *Part* is selected, and click Ok.



Helpful Tips & Tricks

Here are a couple shortcuts and other debugging skills that might be helpful as you go along. Keep this page handy throughout the tutorial.

1. Escape Key

The escape key up on the top left of your keyboard will become your best friend. If ever you triggered a command you didn't mean to, pushing escape will cancel the command.

Ex: You finished drawing a contour or centerline, but it seems to want to keep drawing.

Pushing 'Esc' will keep what you've already drawn, while getting you out of the drawing mode.

2. Ctrl+Z: How to undo a mess

If you don't want to bother with re-dimensioning something, finding the convoluted way to delete a sketch or are just fed up and think it's easier to just do the whole step over again, Ctrl+Z will undo the last step. If you Ctrl+Z too many times, use Ctrl+Y to redo.

3. Click & Drag to select parts of a sketch

Helpful for selecting multiple parts of a sketch. Instead of holding control while clicking each line segment, you can draw a selection box by holding down the left mouse button and dragging it across the screen. The place you clicked and the location of your mouse will form two opposite corners of the selection box.

Note: this will only work to select things in the current sketch you are editing. This will not highlight previous sketches or features.

4. Feature vs. Sketch: Extrude out? Extrude cut? or Oops, I forgot to select a feature!

A sketch is a 2D representation, like a sketch on paper. A feature is what is added to a sketch to make it 3D. Note that a sketch always precedes a feature.

Step 1: Removing an incorrect feature from a correct sketch

Right click on the incorrect feature, choose delete. The sketch should still be there.

Step 2: Your sketch is awesome and complete, but there's no feature attached!

Click on the sketch name in the left panel, and then click the correct feature button.

5. Rotating the component

Holding down the middle mouse button while moving the mouse around allows for viewing the part at any angle. This can be useful for amusing yourself as well as other more practical purposes.

Ex: When in *Features*, editing the extrusions, SolidWorks will put in a phantom projection of what it will look like. You can view it by rotating your piece, checking if the proportions and locations of the extrusions/cuts are visually correct.

6. Editing Smart Dimensions

When in sketch mode, you can change an already placed dimension by double clicking on the dimension. This only works if you are not in *Smart Dimension* mode (use Esc). It is not necessary to delete the dimension and reenter it.

7. Zooming

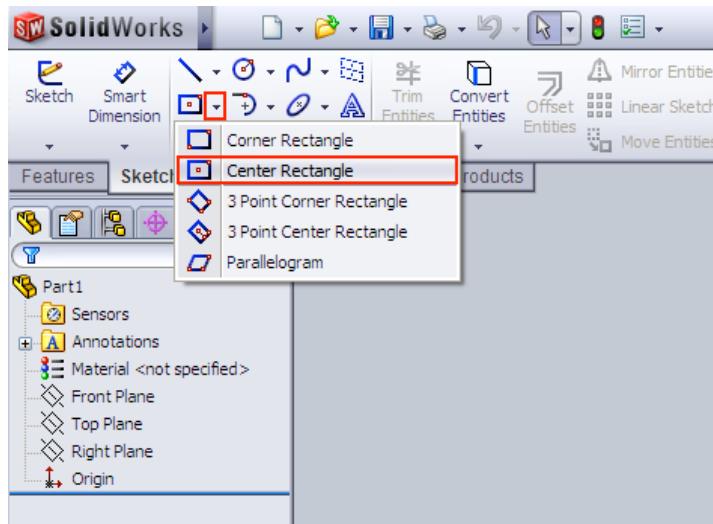
To zoom in and out, use the scroll (middle mouse button). It will zoom in/out centered around the location of your mouse. Or, you can press the 'f' key to make the part 'fit the screen'.

8. Save Often

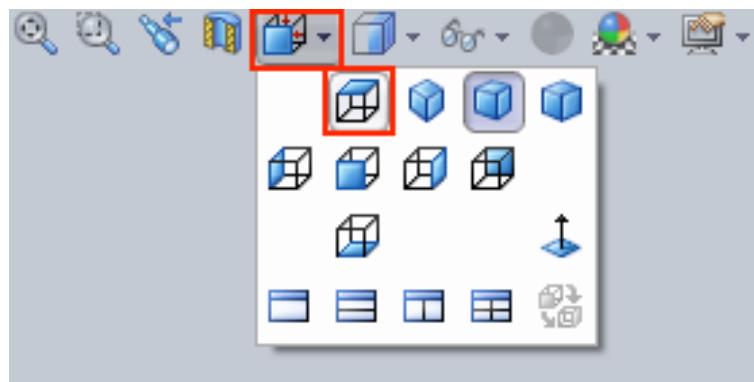
To save your work, click on the Save icon at the top of the screen or press Ctrl-S.

Starting the base

Now, we'll get started with drawing the part. We want the base to be a single rectangle, centered on the origin. In the Sketch tab, click on the arrow next to the rectangle, and select *Center Rectangle*.



A set of planes should show up in the main window. Click on *Top Plane*. This means you'll be seeing the part from above. At this point, you should change perspective to make drawing more intuitive. There is a button on top of the main window that lets you select a perspective- this is the View Orientation button. Use it to shift into downward-looking viewpoint.

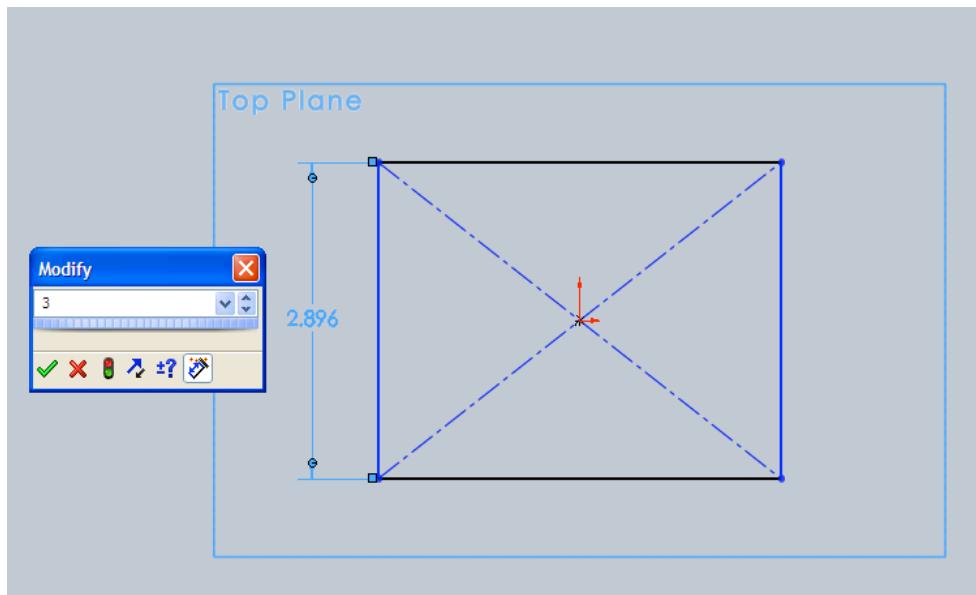


Now, click in the very center of the plane (the cursor should show an orange dot) and draw a rectangle- the exact size does *not* matter.

All of the lines defining this rectangle are blue. That's because the rectangle doesn't yet have any associated dimensions. To assign dimensions, click on *Smart Dimension* in the upper corner of the *Sketch* tab.

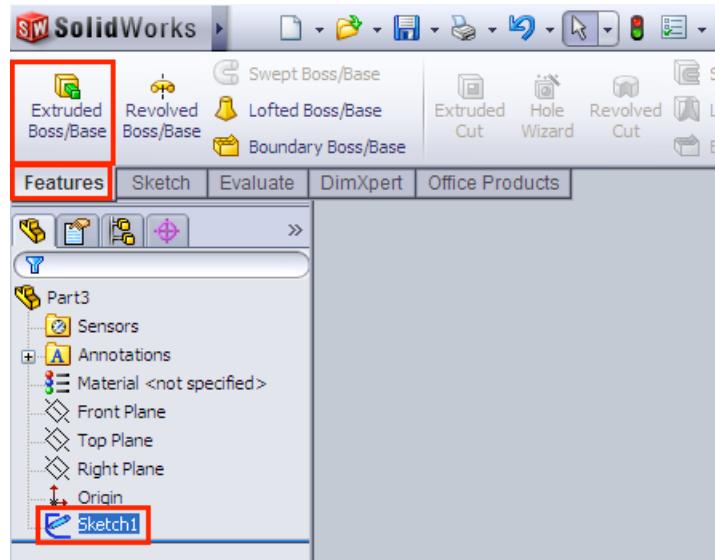


Click on the rectangle's left side, move the cursor a bit to the left, and click again to place the dimension on the sketch. At this point, a window should pop up allow you to set the dimension. We want the rectangle to be 3" by 5", so enter 3 in the window and click the check box.

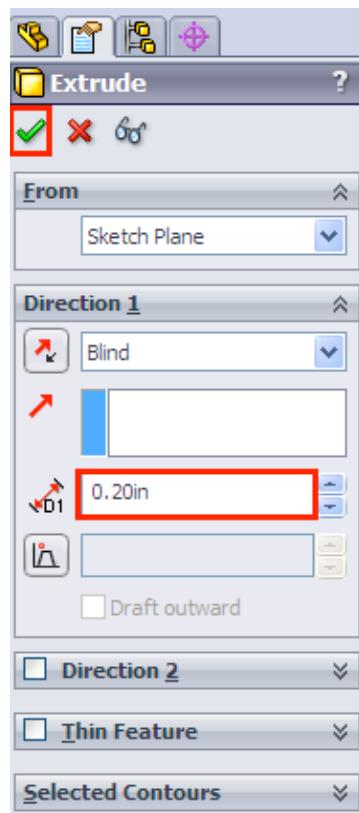


Now, repeat the process for the long side, making it 5" long. All the lines of the rectangle should now be black, and you should see "Fully Defined" in the status bar at the bottom of the screen. We're all done with the sketch, so click on *Exit Sketch* in the upper left corner of the Sketch tab.

Now, we're going to push the sketch into the 3rd dimension! Switch to the Features tab, and select *Extruded Boss/Base* with Sketch1 selected.



Select a depth of 0.2 inches, and click on the green check mark.



You should now see a rectangular prism in the main window. Hold down the middle mouse button to spin around the part; press 'f' to zoom to the extents of your part.

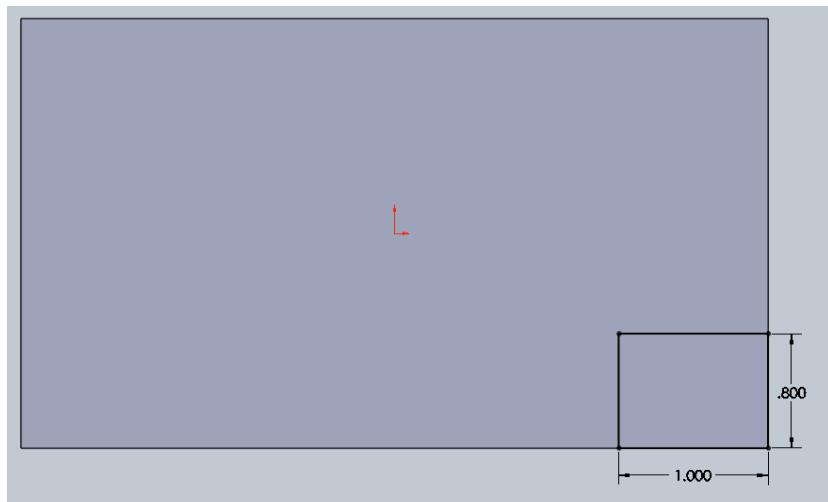
Next, we will cut out excess material on the front of the robot. In the Features tab, select *Extruded Cut*. Note: **do not** choose *Extruded Boss/Base*.

SolidWorks now asks for a plane on which to draw- this is what's known as the sketch plane. Click the top face of the rectangular prism.

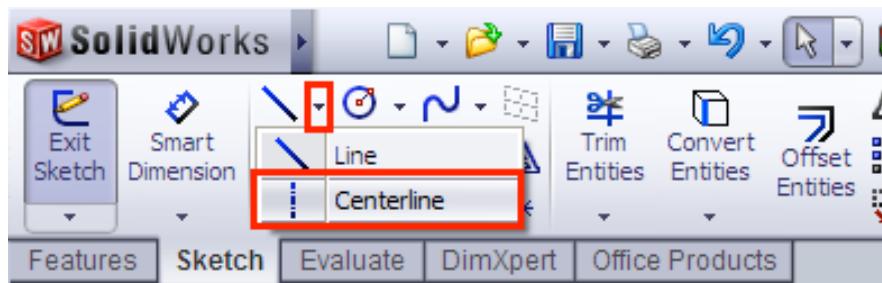
Note: If you've gotten disoriented from all your spinning, remember that you can reorient yourself using the View Orientation button at the top of the window to return to the top view.

This time, select a *Corner Rectangle* from the Sketch tab. Draw a rectangle from the bottom right corner up. SolidWorks is enthusiastic about snapping sketches to existing lines and corners in the drawing. Visual cues should indicate when the rectangle is snapping to the corner of the base. Again, the exact size doesn't matter, since it will change when you apply dimensions to the rectangle.

Use Smart Dimension to define the rectangle as 1" by 0.8". You should end up with the following:

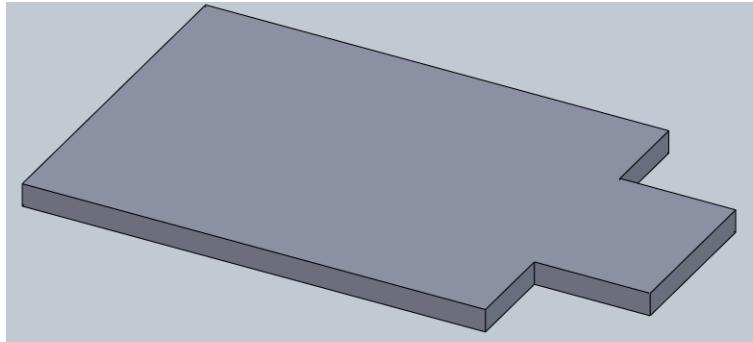


To save on effort, we will now mirror this rectangle along the long axis of the base. Select *Centerline* in the Sketch tab. This tool will be very useful for making symmetric parts.



The centerline should connect the midpoints of the left and right sides of the base. Move the mouse along the left side of the base. When you are near the middle, an orange square should appear, indicating the midpoint of the line. Click on the square to start the centerline, move horizontally to the right side's midpoint, and click again to place a vertex. Press Esc once or Enter twice to complete the line.

Select the rectangle that you've just drawn and the mirror line (either hold the Ctrl key and click multiple times or Click & Drag to select multiple items - refer to Helpful Tips & Tricks). Now, click *Mirror Entities* in the Sketch tab. SolidWorks will automatically decide that you want to mirror about the centerline. It should reflect the rectangle onto the upper half of the base. Now, exit the sketch. A new options menu for the Extrude command should appear on the left hand side of the screen. In the options for Extrude, change "Blind" to "Through All". This will ensure that if you change the thickness of the base, the cutouts will still go through the entire thing. Click the green checkmark, and admire your work.

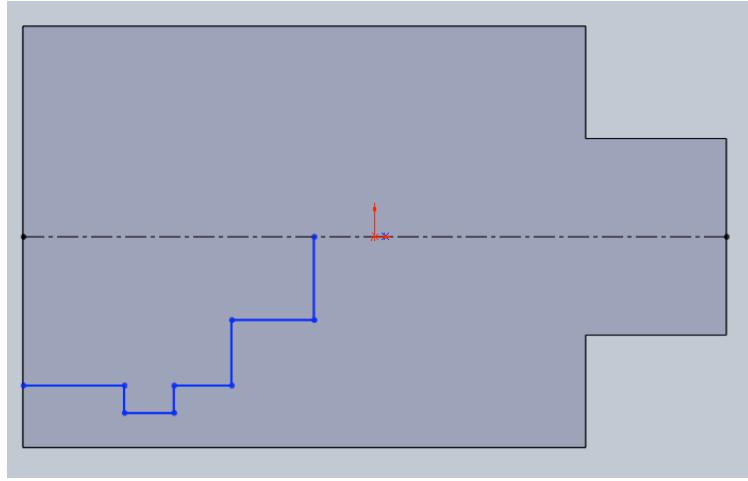


Believe it or not, the basics that you've just learned will allow you to make all kinds of complex parts. At this point, let's take a break and rename the steps that you've taken. In the panel on the left side of the screen, you'll see "Extrude 1" and "Extrude 2". These aren't very descriptive names! Click on the text twice to change it. (Don't double-click; wait a moment between the consecutive clicks.) "Base" and "Front cutouts" are better names for these steps. When keeping track of a complex part, descriptive names save a lot of time! If you ever need to go back to correct a step, you can click on that step and adjust dimensions as necessary, or even delete the entire step. You may also add new corrected steps and rearrange the order of steps. We won't keep reminding you, but as you go through this tutorial, it's a good idea to change the default names after each step.

This is a great time to save your work. Save again at regular intervals as you go.

Cutouts for gearbox and motors

It's time to add an inset for the gearbox and motors. Again, select *Extruded Cut* and click on the top face of the base. First, draw a centerline across the base. Then, in the Sketch tab, select *Line* and draw something like the following:

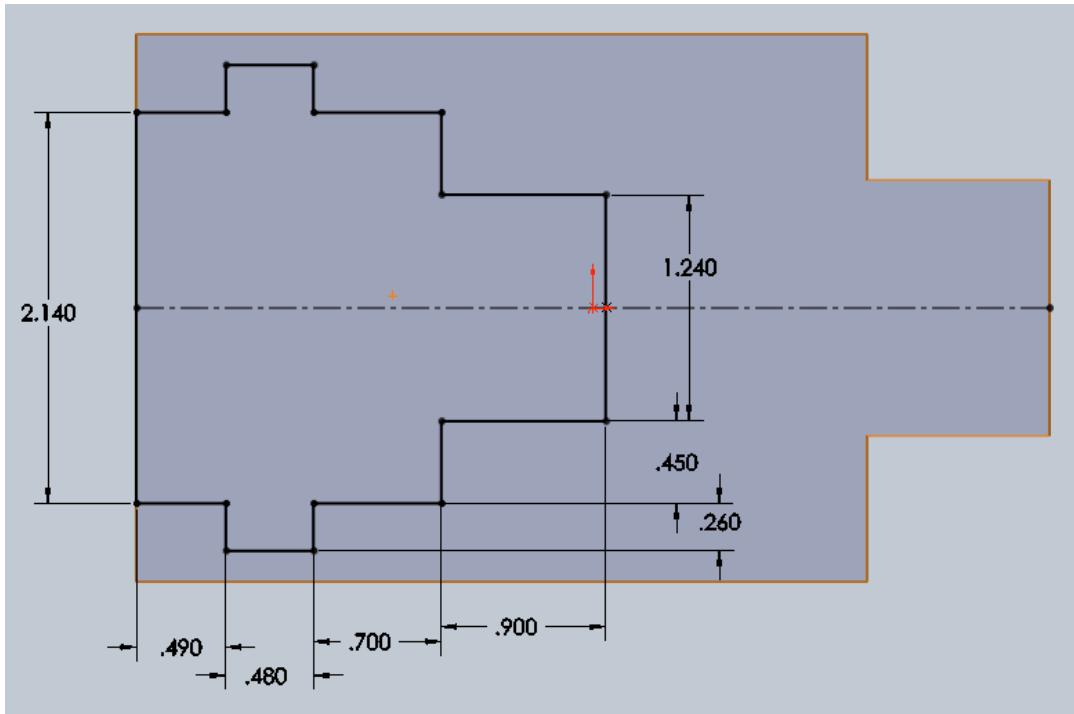


Be careful to keep that the lines stay either horizontal or vertical. Make sure that the line starts on the side of the base, and ends on the centerline. SolidWorks gives you visual cues (e.g. changing colors) to show when it's snapping points to other lines on the drawing.

We want this shape on both sides of the centerline. Again, select all of the lines and the centerline; press *Mirror Entities* in the Sketch tab to make a reflection. Make sure that the contour is closed- if there isn't one already, draw a line from the top left to bottom left vertex.

Now, use *Smart Dimension* to give this shape dimensions. Observe that as you set dimensions on the bottom of the sketch, the top half adjusts itself to stay mirrored. Make sure the piece is fully defined.

Note: You do *not* want to over define the sketch. If there is a warning box asking if you want to make a dimension 'driven', that indicates the dimension you are trying to add is unnecessary.

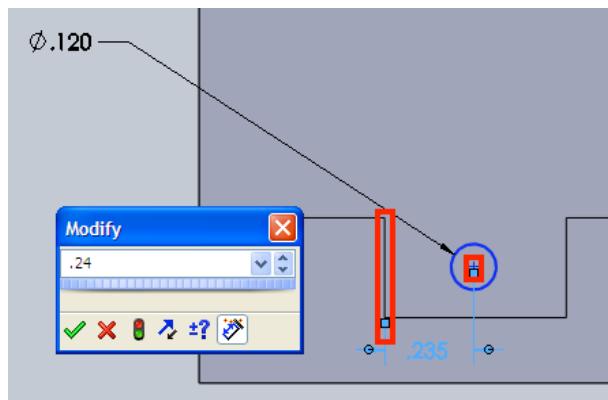


Now, exit the sketch. Set the extrusion's end condition to "Blind", and pick a depth of 0.1". This will cut part way through the body. Complete the extrusion, and look over your work.

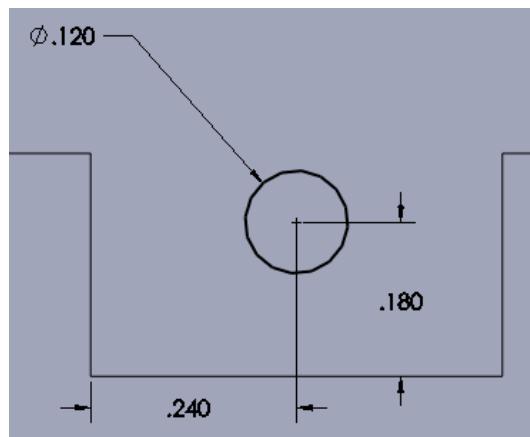
Satisfied? Good!

The gearbox has screw holes that make it easy to attach. We can punch holes through the base for these screws. Select *Extruded Cut*, with the gearbox inset as your sketch plane. Select *Circle* from the sketch tab, and draw a circle inside the tab at the bottom of the cutout. As always, exact dimensions don't matter at this stage.

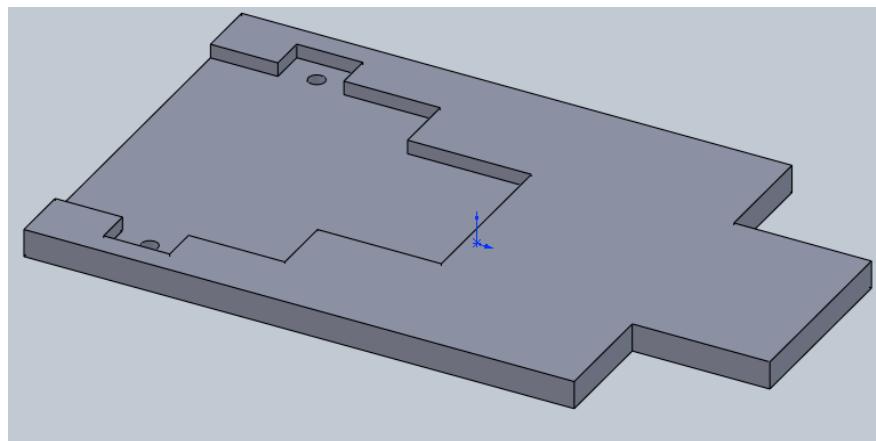
Using *Smart Dimension*, click on the outside of the circle. You'll create a diameter dimension; set it to 0.12". The circle's size is now defined, but you still need to give it a location. Click on the center of the circle, then on the line on the left side of the tab:



This sets the perpendicular distance from the edge to the center. You want this value to be 0.24". Similarly, set the distance between the circle's center and the bottom of the inset at 0.18". The final set of dimensions should look like the following:

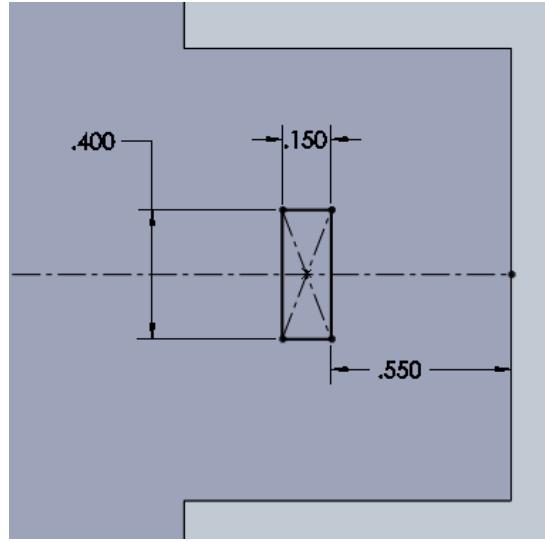


As before, mirror this circle about a centerline, then exit the sketch. Set the extrusion mode to "Through All", and make sure that everything looks correct:

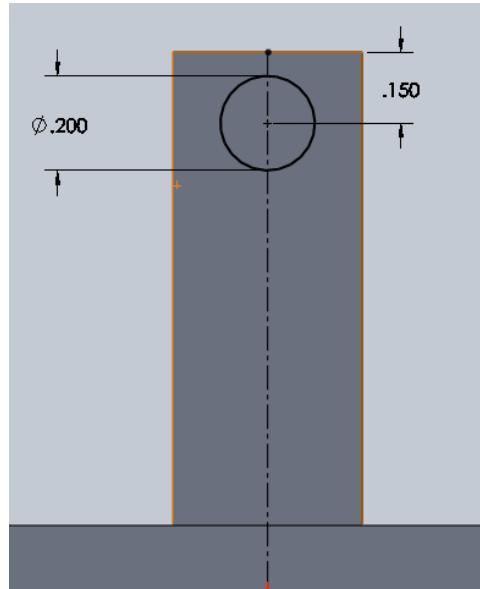


Phototransistor tower

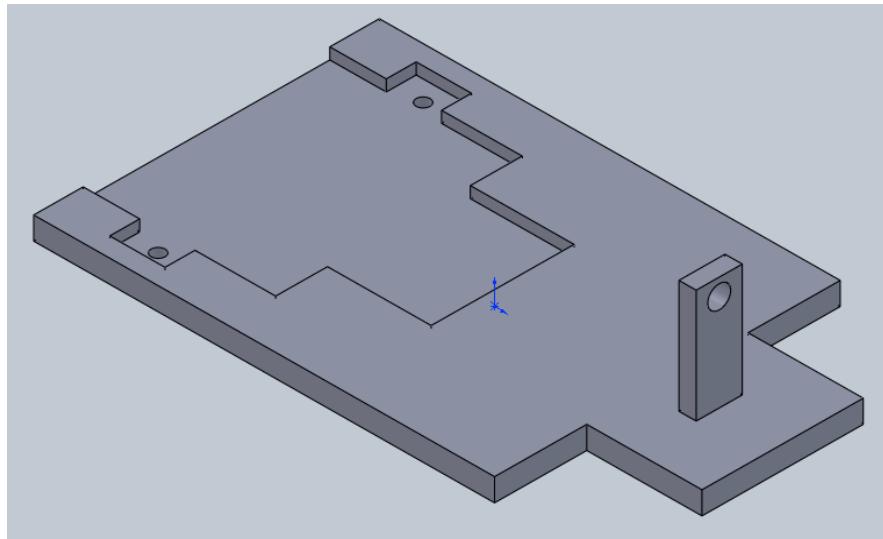
There are two steps remaining in the guided part of this lab: extrude a tower for the phototransistor, then cut a hole in it. This tower will go on the front of the robot. To start, select *Extruded Boss/Base* from the Features tab. Select the top level of the base as your sketch plane. As usual, it will be helpful to put down a centerline. Draw a *Center Rectangle* on the centerline, then apply the following set of dimensions:



Extrude the tower up by 1". The phototransistor will rest inside this tower, but it needs a hole in which to sit. Start making an *Extruded Cut*, and select the front of the tower as the sketch plane. At this point, it might be helpful to adjust your view so that you're looking at the front of the chassis. Draw a centerline down the tower, and draw the following circle:



Now exit the sketch, and select "Up to Next" as the extrusion type. This will cause the hole to be cut through the tower, but not any farther. At this point, your chassis should look like the following:



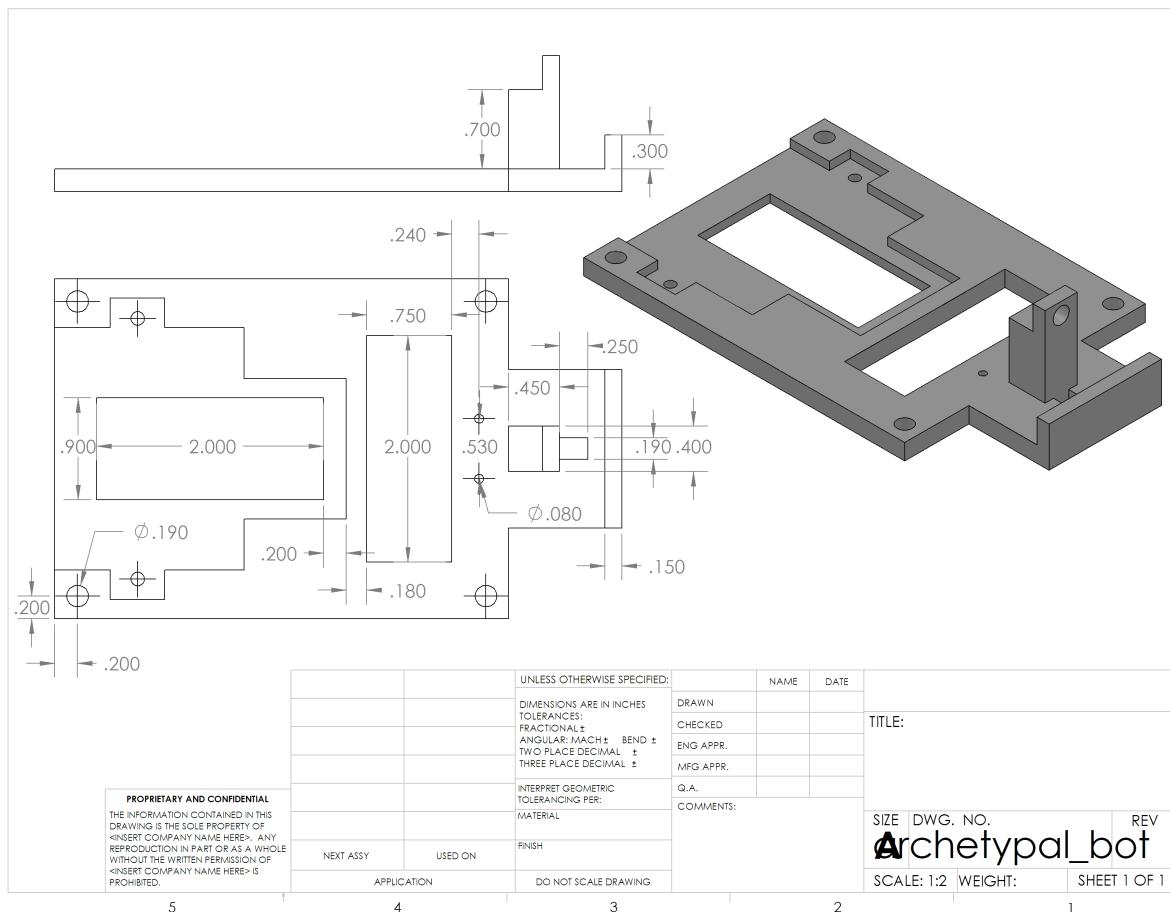
Identification Label

Put your name on the bottom side of the robot chassis so you'll be able to identify your robot after it is manufactured. Use an extruded cut. Choose the text tool. Enter your initials. The default font isn't very readable. Uncheck the "Use Document Font" box and click on Font. Choose Arial Black and set the height to 0.2 inches. Apply dimensions to place it in a good location. Set the depth to 0.05 inches.

Boldly onwards!

Now, look at the drawing of the completed chassis. You will see that you are missing various important pieces:

- Corner holes for attaching PCB
- Square hole for the reflectance sensor
- Pair of holes for attaching caster wheel
- Support structure for phototransistor tower
- Front bumping plate
- Miscellaneous cutouts to reduce part volume



The drawing has enough information for you to add these pieces to your own chassis. Proceed boldly onwards, and ask your instructor for clarification if you have any questions.

When you have finished, check how much material your chassis will use. This can be found in the toolbar that slides out when you hover your cursor on the SolidWorks logo in the upper left corner. Try Tools --> Mass Properties. The standard bot should have a volume of 1.87 in³, minus a very small amount (e.g. 0.01 in³) due to the initials. If this is correct, ask your instructor to check off your completed drawing.

Preparing for 3D printing

Finally, write out your design to a printable file format. In order to see the upper menus, hover the cursor on the SolidWorks logo in the upper left corner. Go to File -> Save As. Change “Save as Type” to STL (a *stereolithography* file format), and name the file “username_bot.stl” (i.e. Ben Bitdiddle’s bot would be named “bbitdiddle_bot.stl”). That’s all you need to do to make your chassis ready for the 3D printer!

Email your .stl file to the ECF Systems Administrator, willie_drake@hmc.edu. In the subject line, indicate “E11 3D print request for <username>” with your username.

If you wish to print designs in the future for your coursework or for personal projects, contact Willie Drake for a 3D Printer Request form. Complete the form and send in the file. You are responsible for the cost (\$10/in³) of future designs you print after this laboratory. Bring a check made out to Harvey Mudd College to Cynthia Wheeler in the Engineering Department Office.