LABORATORY ASSIGNMENT NUMBER 2 FOR CMPE 118

Due by 6:00pm on Thursday, January 31, 2008
Pre-Lab Due by 5:00pm on Friday, January 25, 2008

Purpose:
This lab is intended to acquaint you with:
- Using SolidWorks as a visualization tool.
- Developing parts in SolidWorks/CorelDraw
- Cutting parts using the Laser Cutter
- Working in Foamcore
- Assembling a working filter

Minimum Parts Required:
There is a selection of motors set out in the lab that you may use as the motors to develop your motor mounts. The motors are just for use as props in this lab. I want them back. Foamcore, 3/16” MDF, hot-glue guns, hot-glue sticks, eXacto knives, lots of blades.

For the circuit part, a small perf board (from BELS) and soldering station (provided) and/or wire-wrap tools.

Note: In order to save on material costs, you should team up with AT LEAST one other person on this lab (in order to get to know you classmates, team up with someone you have not worked with before). You are responsible for providing your own materials.

Pre-Lab:

Complete the following exercises AFTER you have read through the lab assignment and BEFORE starting to work on the parts of the lab.

0.1) Set yourself up at any workstation that has SolidWorks installed (all the machines in BE115 do) and follow the SolidWorks design tutorial available at:
http://www.me.cmu.edu/academics/courses/NSF_Edu_Proj/Statics_Solidworks/index.htm

0.2) Look through the attached pages on the PRL SolidEdge Tutorial – NOTE: We use SolidWorks, not SolidEdge, but you will be building this part in SolidWorks for the prelab, so it is useful to see how it goes together. Watch the SolidWorks tutorial lecture at:
http://www.soe.ucsc.edu/classes/cmpe118/Winter06/Videos/SolidWorksDemo.avi

In the report:
Include a dimensioned three view and shaded printout of the part from the tutorial in 0.2.

Part 1 Designing A Simple Motorized Platform

Reading:
Fabulous Foamcore. (on the website)

Assignment:
You are to design, capture the design and assemble a simple motorized platform. The platform should have a flat base made from two layers of Foamcore and it should carry two DC gear-motors and an H-Bridge Module (2” x 2” electronic part). The motors should be mounted to the base using motor mounts constructed of Foamcore. The mounts should attach to the base using ‘Tab in Slot’ construction. The motor mounts should provide more robust support than the simple planar design shown in class. The H-Bridge Module should be attached to the top of the base near the motors. Also mounted to the platform should be a 6” diameter 8” tall circular column, constructed of foamcore and centered on the base. Sitting atop the column should be a smaller platform, also made of foamcore. The platform should have the shape of a square box of about 1” depth and be centered on the platform.

1.1) Using SolidWorks (or any other drafting program, SketchUp, Visio, CorelDraw, etc.) to construct simple 3-D shapes to represent the base, motors, motor driver board, column and
platform. Create an assembly of these parts to explore how they will fit together.

1.2) Using the Fabulous Foamcore handout, and a sharp eXacto knife (be careful!), build the foamcore box that will sit atop the platform. Use lap joints at the edges.

1.3) As above, build the foamcore column. Use a lap joint to close the column.

1.4) Figure out how you are going to attach the column to the base and to the platform. You may want to do this BEFORE you actually build them.

1.5) Using SolidWorks, create the parts necessary to assemble the motor mounts that you designed and mount them to the base. The finished base should be roughly circular with recessed cutouts to provide room for 4” wheels to be mounted on the motors. You will need to move the 2D shapes to CorelDraw for part 2 of this lab.

1.6) Using SolidWorks, create 3-1/2” wheels to be mounted on the motor shafts. These will need to be at least three layers of foamcore, or two layers of MDF. You will need to move the 2D shapes to CorelDraw for part 2 of this lab.

In the report: Include a printout of the model from part 1.1 and the individual parts from parts 1.5 and 1.6.

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**Part 2 Implementing A Simple Motorized Platform**

**Reading:** CMPE-118 LaserCutter Handout

**Assignment:** Take the design that you created in Part 1 and implement a prototype of the platform.

2.1) Using the laser cutter handout (on the website), to be distributed later, and you printouts from the above drafting part, to prepare your part designs for cutting using the laser cutter. (Note that the laser cutter is driven from CorelDraw, so you will need to get your parts into that program).

**NOTE: No Etching Allowed.** Aside from markings in order to help you assemble parts, there is no etching allowed on your designs. Remember that you will have to baby-sit your design while it is being cut out, and that means you will have to wait.

2.2) Have your output files reviewed by John, Erik or Gabriel.

2.3) Cut the parts from 3/16” MDF (Medium Density Fiberboard), using the Laser Cutter.

2.4) Assemble the parts of the platform. Do not glue. Demonstrate it to John, Erik or Gabriel.

2.5) Fit/glue the parts together (NOT the motors). Demonstrate it to John, Erik or Gabriel.

In the report: Include printouts of the SolidWorks/CorelDraw files that you created to help you cut out the foamcore/MDF.

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**Part 3 Building Your Detector Circuit**

**Reading:** None.

**Assignment:** Take the design that you created in Lab 1 and build a working version that you will use on your final project.
3.1) Make sure you use a circuit that actually works well (talk to your classmates about this, see whose design really worked well from Lab 2, and try to make one like it).

3.2) Have your design reviewed by John, Erik or Gabriel.

3.3) Do NOT disassemble your working one off of your protoboard, instead, replicate the design on the perf board, and solder or wirewrap the parts together.

3.4) Test your assembly and make sure it works, if not, debug. Again, incremental development here; build a little, test a little, build a little more, test a little more, until the whole thing functions reliably.

3.5) Demonstrate it to John, Erik or Gabriel.

In the report: Include a schematic of the final circuit you built, and if you can, add in a digital picture of the final board, top and bottom. If you did a simulation of the circuit, include that too.

Lab #2

Time Summary

Be sure to turn this in with your lab report

This information is being gathered solely to produce statistical information to help improve the lab assignments.

<table>
<thead>
<tr>
<th>Pre-Lab</th>
<th>Preparing Outside of the lab</th>
<th>Preparing the Lab Report</th>
<th>In the lab working this part</th>
<th>In the lab working this part</th>
<th>In the lab working this part</th>
<th>In the lab working this part</th>
</tr>
</thead>
<tbody>
<tr>
<td>Part 1</td>
<td>Preparing Outside of the lab</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Part 2</td>
<td>Preparing Outside of the lab</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Part 3</td>
<td>Preparing Outside of the lab</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Welcome to PRL's Solid Edge Tutorial.

This tutorial is designed to familiarize you with the interface of Solid Edge Part V.9 and many of its useful tools. It teaches how to model and edit a sample part called RingTutorial, which is part of a magnetizing glass assembly.

Sections in this tutorial:
- Open Solid Edge Part
- Get Familiar with Solid Edge Part
- Toolbars
- Creating the Part
- Protrusions
- Sketch mode
- Basic Dimensioning
- View Manipulation
- EdgeBar and Feature PathFinder
- Defining Extent
- Patterning
- Defining the Profile Plane
- Sketching mode
- Creating holes
- More Basic Dimensioning
- Rounds
- Chamfers
- Changing the appearance of the part environment
- Changing color
- Changing the profile plane
- EdgeBar and Feature PathFinder
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- Basic Dimensioning
To dimension the circle:

- Select the Circle by Center tool.
- Move the cursor to the intersection of the axes until the midpoint alignment indicator appears.
- Click once to place the center of the circle.
- Drag the cursor, and click once more to place a circle of any size.

To dimension the circle:

- Click on the SmartDimension tool.
- Place the cursor somewhere on the circle so that the circle highlights.
- Click once to start the dimension, drag the dimension out, and click to place it.
- On the ribbon bar, key in a diameter of 2.416".
This dimension constrains the diameter of the circle. When you originally placed the circle, the alignment indicator ensured placement on the axis intersection. Solid Edge then placed its own constraints, ensuring that the circle is locked into location on those reference planes. These constraints are indicated by a relationship handle in the form shown at right. • Click Finish, this will complete the sketch.

Now that you have completed sketching the profile, Solid Edge wants to know how thick you want the protrusion to be. By dragging the mouse you can dynamically change the thickness. • Key in the distance .375" and press Enter. • Click either above or below the profile to indicate which way to extrude the material. It does not matter which way you extrude. • Click Finish to complete the first feature.

View Manipulation
You have just created the base protrusion. Take this opportunity to get a different view on it. There are rotate, zoom, and pan tools on the Main Toolbar. EdgeBar and the Feature Pathfinder
The EdgeBar is a window that allows you to keep track of your design process. It has several tabs, including the Feature Pathfinder, which allows you unique access to and control over the features you've created in your part.

• Open the EdgeBar with the tool on the Main Toolbar. • Click on the Feature Pathfinder tab at the bottom of the EdgeBar. • Select the Select tool on the Features Toolbar. • In the Feature PathFinder, right click on Protrusion 1, (the protrusion you just created) and select Rename. • Type in the new name for this feature, "Outer Ring." It is a good idea to explore the other tabs on the EdgeBar as well.

Step 2: Create Leg Tab Original Protrusion
• Select Protrusion Tool again. Defining the Profile Plane
At this moment, Solid Edge is asking you to take steps to define the profile plane (the plane in which you will sketch) for the next protrusion. • Click on the top surface of the base protrusion, which selects it as the profile plane. Now, Solid Edge wants to know in what orientation you want to look at this profile plane. In the case of the circle, it doesn't matter, but let's learn the steps anyhow. The instruction line asks you to "Click on the edge, face, or plane to be the base of the profile plane." You may essentially think of this so-called "base of the profile plane" to be the horizon of the new view you will have in sketch mode. • Click on the reference plane as shown in the image on the left. Finally, Solid Edge wants to know which way in your new view will be right side up. It wants you to place the new red rectangle in what you wish to be the lower left-hand corner of your new sketch view. • Click with the rectangle in the location shown above on the right. Now you are once again in sketch mode. • Click on Circle by Center tool. • Move the cursor until Solid Edge indicates that you are at the diameter of the previous protrusion profile and on the Y-axis. • Place the circle, keying in a diameter of .750", and click Finish. • Place a dimension on the circle using the SmartDimension tool, as before.
The leg tab sketch is now completely constrained. By constrained, I mean that its XY position has been specified relative to the diameter of the base protrusion and the Y-axis. The diameter has also been fixed to 0.750".

Defining Extent
Again, we need to define the extent of this protrusion. This time instead of keying in a thickness, let's specify extent by choosing an extent option from the Ribbon Bar.

1. Click on From/To Extent
2. Click once on the top surface of the Outer Ring
3. Click once on the bottom surface of the Outer Ring
4. Click Finish

Solid Edge completes the feature for you.

• Rename this feature "Leg Tab Original" in the Feature PathFinder.

Step 3: Create Leg Tab Pattern
You just created the basis for one leg tab, but the part requires three. Instead of creating two more protrusions, let's just create a pattern of features based on the leg tab original.

Patterning
1. Select the Leg Tab Original feature, either from the part view window or from the Feature PathFinder
2. Select the Pattern tool from the Features Toolbar
3. Select the top surface of the part
4. Define the profile plane as you did with the leg tab original
5. In profile mode, select the Circular Pattern tool
6. Click at the center of the large circle, then drag the new pattern definition circle until it matches the original protrusion size. Click to place the circle.
7. On the Ribbon Bar, specify the following options: Fit, Full Circle, and Count = 3
8. Click Finish.

Here, Solid Edge shows you a preview of the leg tab pattern, and offers the opportunity for you to edit any step in the process of its creation.

1. Click the "Smart" pattern option on the Ribbon Bar (it causes Solid Edge to interpolate relationships and is a safer choice for most features)
2. Click Finish
3. Rename the feature "Leg Tab Pattern" on the Feature PathFinder

TIP:
It is always a good practice to constrain every sketch you make. A correctly constrained sketch will make editing your part much easier.

Step 4: Create Lens Hole Cutout
Cutouts
1. Select the Cutout tool from the Features Toolbar
2. Select the top surface of the part and define the sketch plane as before
3. In sketch mode, draw a circle, centered on the part, with any diameter.
4. Place a dimension on the circle, and key in 1.670".
5. Finish the sketch
6. Define the extent of the cutout, this time choosing "Through All" as your extent.
7. Pick the arrow direction that indicates cutting THROUGH the part (down)
8. When you are satisfied, finish the feature and rename it "Lens Hole"

Save your part.
Let's pause for a minute and play.

Changing the Appearance of the Part Environment
Go to Tools -> Options -> Colors tab
On this menu, you can change the colors for all the aspects of your part environment. Just for kicks, change your part color to bright green and your background color to magenta. Now change it to something else you'd prefer to look at.

Step 5: Create the Fastener Holes
On the part drawing, notice the three countersunk holes for 8-32 fasteners. As with any feature, there are several ways to model these holes. You could modify the original leg tab protrusion to include a countersunk hole, then create another protrusion, then pattern that hole as a new feature, then pattern that hole by itself. Or you could create all three individual holes. So that we can do these holes in one feature, let's choose the last option.

Holes
1. Choose the Hole tool from the Features Toolbar
2. Define your profile plane as usual, and enter the profile step
3. Click the Hole Options button on the Ribbon Bar
4. The Hole Options button brings up Hole Settings, where you can choose all the characteristics of the hole. Here we will mandate how the hole is countersunk and that it be suitable as clearance for an 8-32 fastener.
5. Choose Countersunk Hole and key in:
   - Hole diameter: .164
   - Sink diameter: .375
   - Sink angle: 82°
Step 6: Create Rounds

Rounds

- Select the Round tool from the Features Toolbar
- Select the edges shown highlighted below, and key in the radius .375
- Click the green checkmark (or the red X if you need to clear selections)
- Preview and Finish the feature when you are satisfied.

Step 7: Create Chamfers

Chamfers

- Select the Chamfer tool from the Round tool's Flyout
- Choose the top outer and inner edges to be the chamfered edges
- Key in a .100 setback (Default chamfer is 45°)
- Finish the feature

At this point, the part is done and should look like the image on the following page.

Save the Part.

Now let's go back and edit it:

Editing Features

1. Go to the Part

2. Go to the edit

3. Go to the profile button

4. Go to the edit button

5. Key in the diameter of the outer ring

6. Finish the feature

At this point, the part is done and should look like the image on the following page.

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- Finish the feature

At this point, the part is done and should look like the image on the following page.

Save the Part.
• Select the diameter of the Outer Ring
You are now able to edit the diameter of the outer ring.

• Key in a dimension of 4.00"

Keep clicking Finish until you are out of the edit environment and the part regenerates.

Notice how Solid Edge regenerates all your other features to keep the part consistent with the original intention.

What was the original intention???
Remember that the first two holes were constrained to the center of the Leg Tabs and that the Leg Tabs were constrained to the diameter of the Outer Ring. That is why those features were moved outward when the Outer Ring was changed.

The third hole was dimensioned to the reference planes, which are constant and will never move in Solid Edge. This means the third hole is constrained to the reference planes, which were moved outward when the center Leg Tabs were moved outward. The third hole is locked at its position no matter how the part changes.

Now edit the part and change the diameter back to 2.416". Try to edit other features and get used to how to use the dynamic Ribbon Bar to access various aspects of the part.

Congratulations! You have completely modeled the part in Solid Edge!
Welcome to SPDL/PRL's Solid Edge Tutorial.

This tutorial is designed to familiarize you with the interface of Solid Edge Assembly V.9 and many of its useful tools. It teaches how to model and edit a sample assembly called MagnifyingGlass.asm, from the ring you already produced and three included files called leg, lens, and screw.

Sections in this tutorial:
- Creating the Assembly
- Edge Bar
- Orientation
- Mating Surfaces
- Axial Alignment
- Patterning Parts
- Aligning Planes
- Checking for Interference
- Moving Underdefined Parts
- Editing Relationships
- Assigning Constraints
- Checking for Interference
- Cleaning the Assembly
Creating the Assembly

Step 1: Insert the Ring

- Open Solid Edge Assembly from the Programs list
- On the main toolbar, select the Edgebar tool.
- At the bottom of the Edge Bar window, select the Parts Library Tab.
- On the Parts Library tab, click the browse for folder button.

The default location of the Magnifying Glass part files: _Samples>Solid Edge>MGParts on EE118-Server.

(Tip: From explorer, right-click on the shortcut icon to display the shortcut menu. On the shortcut menu, click Hide All Reference Planes. This will clear up the window and allow for easier part positioning.)

Orientation

You will now place your first part in the assembly.
- Double click on ring.par.

This will insert the ring you previously created into the assembly environment. Its position will be fixed (grounded) and it will be the reference for all future parts in the assembly.

Tip: The first part you place into an assembly is important. It serves as the foundation upon which the rest of the assembly will be built. It will remain fixed in the original position and orientation during the lifetime of the assembly.

Step 2: Placing the Lens in the Assembly

- Double click on lens.par
- On the Ribbon Bar, the Mate command will be active by default.

The Mate command simply positions the face of one part towards the face of another part. The Mate command is one of six relationship types offered. The others are Planar Align, Axial Align, Insert, Connect, Angle, Tangent and Flash Fit. The ones most used for positioning parts are Mate, Planar Align, and Axial Align.

Selecting the Mating Surface

At this moment, Solid Edge wants to know which face on the lens you are trying to position.
- Click on the shoulder of the lens, shown in red.

Now, Solid Edge will bring you back to the assembly. It is looking for the target part or more simply for the part you want to attach the lens to.
- Click on the ring.

Now, Solid Edge wants to know the face of the ring that the shoulder of the lens should mate to.
- Click on the bottom face of the ring.
- Make sure the fixed offset icon is selected and the offset value is set to 0.00. This is the distance by which the faces will be separated. We want no separation.
- Click OK

You now successfully mated the two faces although you will notice that the lens is still not in the desired position. This is a good indication that we still need to continue adding relationships.
To finish positioning the lens, we need to align the cylindrical face of the lens to the inner cylindrical face of the ring. This will require that we align the axes of the two parts.

• Click on the Axial Align command in the relationship types list. Solid Edge wants to know which axis on the lens you want to align.
• Select the outer ring of the lens.
• Place your cursor over the ring until it is entirely highlighted then select it (this is the target).

Now we need to select the axis on the ring that we are aligning the lens too.
• Place your cursor over the inner surface of the ring and select it when it becomes highlighted.
• To fix the rotational orientation of the lens, select the Fixed Offset tool.
• Click OK.

You have now successfully aligned the lens to the ring, but let’s change the colors around a little bit so you can view the parts a little easier.

Go to Tools -> All Parts Same Color (deselect this)
• Select the lens and then the down arrow to change the default color. Pick something sassy.

Note: The lens could have been positioned using the Insert command, but that command is simply a combination of a Mate and Axial Align command.

Success! You have positioned the lens within the assembly.

Step 3: Attaching the Legs

We are now ready to start placing the legs into the assembly. There are several ways to attach all three legs. You could position each leg into the assembly one by one using the same techniques we just used to place the lens. Or you could position one leg and use the pattern tool to quickly copy the leg into the other two positions. So that we can place the legs in fewer steps, let’s choose the latter.

• Double Click on leg.par in the parts library.
• The mate command should be active by default in the relationship types list. If it is not, then select it.

1. Select the top surface of the leg.
2. Select the ring.
3. Select the bottom surface of the ring.
• Make sure the offset value is set to 0.00 in the Ribbon Bar.
• Click OK.

You will now want to align the axis of the leg to the axis of the hole in the ring. Since there are quite a number of selectable faces in the area, it is helpful to use the Quick Pick tool.
• Click on the axial align command in the relationship types list.
• Position the mouse cursor over the outer cylindrical surfaces of the leg, stop moving the mouse for a moment, and notice that three dots appear next to the cursor.
• Click, and the QuickPick tool will be displayed. Move the cursor over the different boxes on QuickPick, and notice that different cylinders of the model highlight. You can select any of the outer cylindrical faces since they all share a common axis.

Click OK.
Step 4: Placing the Screws

Aligning Planes

The Planar Align command is very similar to the Mate command except that the faces you are selecting to align will be positioned facing the same direction (not towards each other like when you positioned the lens).

To position the screw we will start by aligning the top face of the screw to the top face of the ring.

Click on the top face of the screw. Select the ring as the part to which the screw will attach. Make sure fixed offset icon is selected and the offset is 0.00 then Click OK.

Now that the faces are aligned, we can use our previous tools to finish positioning the screw.

Use the Axial Align command to align the shaft of the screw to the hole in the ring.

Use the Pattern Parts command to add two more screws to the assembly.

Success!! You have positioned the screws within the assembly!!
Let's remove one of the assembly relationships of the leg and see what that does to our model. We will remove the mate on the top of the leg to the bottom surface of the ring.

1. Select the first leg you put into the assembly, leg.par:1, by simply clicking on it in the assembly window.
2. Click Edit on the Ribbon Bar.
   - The Ribbon Bar should have changed to display the steps you went through to create the assembly relationships.
3. Select the mate relationship in the Relationship List.
4. Click Delete on your keyboard.
5. Click Esc on your keyboard.

Nothing in the graphics window should have changed so let's play around with the position of the leg now that it is underdefined.

1. Click Move Part in the Features toolbar.
2. Select the leg.
3. Put your mouse over the z-axis and click to drag the leg.
   - Notice how the leg is only free to move in the z direction.
4. Select the Rotate command.
5. Put your mouse over any axis and click to rotate the leg.
   - The leg should not rotate about any axis since the axial align command is still active.
6. Click Esc on the keyboard.

### Checking for Interference

Now let's take a look at the interference profile between the Leg and the Ring. This is a simple process where you select the two parts that you want to check.

1. Make sure the leg is imbedded in the ring.
2. Tools -> Check Interference…
3. Select the Leg.
4. Click the Accept tool on the Ribbon Bar.
5. Select the Ring.
6. Click Process on the Ribbon Bar.
   - The outline in red is the exact interference geometry.
7. Click Esc on the keyboard.

Now let's get it back to normal.

1. Select the leg.
2. Click Edit on the Ribbon Bar.
3. Make sure the Mate Command is selected.
4. Mate the top of the leg to the bottom plane of the ring.
5. Click OK.
   - The legs should now be back in their original position.

Now change the colors around a little bit so that you are happy with the way it looks.

TIP: A simple check to ensure everything is correct when you have completed an assembly is to use the interference tool to make sure the parts are not overlapping.
Congratulations!
You have completely modeled the assembly in Solid Edge!