SolidWorks Windmill Project

Preparatory Vocational Training and Advanced Vocational Training
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In this exercise, we will make a simple aluminum sheet. You will get acquainted with Sheet Metal as a function in SolidWorks. And, of course, you will make a working drawing of this piece. The lesson will also present SolidWorks SustainabilityXpress.

Work Plan

You will make the base sheet according to the drawing below.

1. First, draw a hexagon. (Polygon)
2. Sketch a circle starting from the center of the hexagon.
3. Next, make an opening at the bottom of the circle and the hexagon.
4. After this, draw a pattern of 3 holes.
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5 Finally, make a working drawing for use in the workshop.

1 Launch SolidWorks and open a new part document.

2 Set the Units to MMGS.

3 Select the Front Plane. Click on Sketch in the Sketch tab of the CommandManager. You do this to activate the Sketch environment.

4 The base sheet is a regular hexagon. Click on Polygon in the CommandManager. In this exercise, we will draw a regular hexagon.
5. Draw a hexagon from the origin.
   1. For the first point of the hexagon, click the origin.
   2. For the second point, click at an arbitrary distance to the right of the origin.
   Be sure that your cursor is horizontal to the origin as indicated by the icon.

6. Set the following settings in the PropertyManager.
   1. The number of sides is set to 6.
   2. Select the option circumscribed circle.
   3. Click on OK.

7. Set the diameter of the circle to Ø230.94 mm using Smart Dimension.
   Because of the size and positioning of the circle, the sides of the hexagon will be 200 mm.
8. Draw and dimension a **Ø100 mm circle** from the origin.

9. Now draw a **Line** from the bottom of the hexagon straight up to the circle. Dimension this line as shown in the figure. The distance between the line and the center of the hexagon is **40 mm**.
Draw a **Centerline** from the origin straight up to the top of the hexagon.

1. Click the function **Centerline**.
2. For the first point, click the **origin**.
3. Then, draw a **Centerline** straight to the top of the hexagon as shown in the figure.

We will now mirror the line.

Select **Mirror Entities**.

The **Options** tab in the **PropertyManager** shows the selected components you want to mirror.

1. First, select the line
2. The window **Entities to mirror** displays the selected component, in this case, the line.
3. Click in the window: **Mirror about**.
4. Select the **Centerline**. Once you have selected the **Centerline**, it is displayed in the **Mirror about** window.
5. Click on **OK**.
13 We want to remove the bottom of the circle and the hexagon to make a recess there.
These areas can be removed with the Trim Entities command.
In the CommandManager, click Trim Entities.

14 In the PropertyManager, select the option Trim to closest.
Now, cut off the bottom part from the hexagon and circle.
In the image on the right, the bottom part of the hexagon has already been removed.
Click on OK.

15 Now, draw a circle.
The center of the circle must be on the centerline as seen in the illustration.
16 Enter the circle dimensions. The circle has a diameter of Ø3.2 mm and is a distance of 76 mm as measured from the origin.

17 We want to add two more circles to the sketch. To do this, use the command **Circular Sketch Pattern**.

1. First, click the arrow to expand the menu under **Linear Sketch Pattern**.
2. Select **Circular Sketch Pattern**.

18 Now, do the following:

1. Set the number of copies to 3.
2. Check **Equal Spacing**.
3. Click anywhere in the window **Entities to Pattern**. The window is empty at first, but as soon as you click the components to be copied, the window displayed the selected components.
4. Select the Ø3.2 mm circle.
5. Click **OK**.

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Tip: You have just found out that using Linear Sketch Pattern or Circular Sketch Pattern will considerably reduce your drawing time. You can easily add objects (lines, circles, rectangles, etc.) according to a specific pattern.

19 Using the **Smart Dimension** function, dimension the circles as shown in the illustration.

20 The sketch still has not been completely defined yet. Adding dimensioning and/or relations provide a completely defined sketch. In this case, you want to add a relation.

1  Click the arrow under **Display/Delete Relations**.

2  Then choose **Add Relations**.
21 Select the center of the two lower circles. The plus sign shows you will be selecting the center of the circle. The selected objects will appear in the blue window, in this case, points. Click the Horizontal relation.

Tip: The function Add Relation allows you to add various relations to entities. This function can also be started by selecting multiple entities while pressing the <Ctrl> key.

For example, you can make two lines run parallel, or set them square without using the dimensioning tool.

To display and/or remove all existing relations, use the function Display/Delete Relations.

22 First, make sure the Sheet Metal buttons are available. The best way to do this is to add them to the CommandManager.

1 With the right mouse button, click a tab in CommandManager.
2 In the displayed menu, click Sheet Metal.

If Sheet Metal it already checked in your system, continue to step 23.

23 Click the Sheet Metal tab in the CommandManager. Then, click Base Flange/Tab.
24 In the PropertyManager, enter 2 mm as the material thickness.
Leave the rest of the menu unchanged.
Click on OK.

25 Next, we will make the chamfer at the top of the holes.
1 In the CommandManager, click the arrow under Fillet.
2 Click Chamfer.

26 You must now set a few things in the PropertyManager.
1 Select the top edge of all three circles.
2 In the blue area there are three edges displayed.
3 Make sure the option Angle distance is selected. If not, check it.
4 For the chamfer distance, enter 1 mm and 45 deg.
5 Make sure the option Full preview is selected.
6 Once everything has been set correctly, click on OK.
27 Change the material to **1060 Alloy**.

The model is now ready.

In the toolbar, click Save and name the file: Base Sheet.

**SolidWorks SustainabilityXpress**

As a developer/designer, you must take several aspects into account. One of these aspects in the environmental impact of your design. SolidWorks SustainabilityXpress allows you to understand and visualize the environmental impacts of your designs and, if necessary, improve the design.

This includes carbon footprint calculations, ((Footprint) is a measure of unit for CO₂ emissions), and real-time feedback on the product, which measures energy consumption during the production of the model as well as the effects on the air and water during production, enabling you to adapt your design and improve the final values.

In the next steps, you will learn how to use this new function.

28 Start **SustainabilityXpress**

1. Click on the **Evaluate Tab** in the **CommandManager**.
2. Select **SustainabilityXpress**.
29 After launching **SustainabilityXpress**, SolidWorks opens a new window on the right.

30 The window includes six areas:

1. **Material**: Enter the material properties here.

2. **Manufacturing**: Specify where the product will be made and using what process.
3 Use: Specify the region where the product will be used.

4 Transportation: The method used to move the product from where it is manufactured to where it is used.

5 End of Life: The percentage of the product that gets recycled, incinerated, or put in a landfill.

6 Environmental Impacts: Four diagrams are displayed here. They show the environmental impact of production and transportation.

31 Let us now take a more detailed look at how SustainabilityXpress works.

1 In step 27, the material has already been defined as Aluminum 1060 Alloy. The software copied and pasted this automatically.
2 In the Manufacturing window, select Stamped/Formed Sheet Metal as the Process. For the region, select Asia.

3 In the Use window, select Europe as the Region. This indicates that the product will be used in Europe.

4 Under Environmental Impacts 4 diagrams are displaced. They describe the environmental impact of the product.
An important part of SustainabilityXpress is the Environmental Impact window. As a designer, you can find here various types of information on the environmental impact of your product/design.

Double-click an individual impact to display the detailed diagrams. They allow you to quickly review the environmental impact through the life of the product.

Click on the home icon to return to the main display.

E.g. **Carbon Footprint**: CO₂ impact on the environment, e.g. greenhouse gas production.

E.g. **Energy Consumption**: The total amount of energy required to manufacture the product.

E.g. **Air Acidification**: Impact on the air! In particular, contribution to acid rain.

E.g. **Water Eutrophication**: Impact on the water! Resulting in algae growth on coastal waters.

Let us now change the production location to see how the environmental impact changes if the base sheet is not produced in Asia but rather somewhere else, for example in Europe.

Change the **Region** to **Europe**.
34 Now, watch the diagrams. There is a significant difference between the first and second calculations. The emission of Carbon is now lower than in the first calculation.

**Current**, is green which means better than the previous location.

**Previous,** grey represents the first calculation, the previous production location.

35 What happens to these values if we choose a different material for the product?

**SustainabilityXpress** has anticipated that possibility. Instead of having to search through a very long list of materials, you will see a list of similar materials.

Click **Find Similar**.
A number of new values must be entered in the newly opened window. This allows you to request a search in one other type of material only. Alternatively, you can ask the program to search through all materials. You can also specify and change different material properties. In this case, we will only change the tensile strength and yield strength requirements.

1. Select **Aluminum Alloys** as the **Material Class**.
2. Next to **Tensile Strength**, change the condition to greater than >.
3. Next to **Yield Strength**, change the condition to similar to ~.
4. Click on **Find Similar**.

![Image of the SolidWorks material selection screen with arrows pointing to the steps: 1. Select Aluminum Alloys as the Material Class. 2. Next to Tensile Strength, change the condition to greater than >. 3. Next to Yield Strength, change the condition to similar to ~. 4. Click on Find Similar.](image-url)
37 Click the option 1345 Alloy. This is almost the same material as the one you had chosen (1060 Alloy). There is one important difference: the tensile strength is significantly higher.

The diagram section immediately displays the new calculation. It is almost identical to the old one. This is because the materials are almost the same.

You can now do the following: Accept, Edit, or Cancel.

Let us choose Edit because we want to know what will happen if we choose steel instead of aluminum.

38 Choose Steel as the Material Class.

Change the conditions for Tensile Strength and Yield Strength to greater than >.

Click Find Similar.
39 **AISI 1020** is a low carbon machine steel offering good general and structural steel properties. We will choose this type of steel.

Click **AISI 1020**.

The diagram section shows this choice is better for the environment, except for water.

Click **Accept**.

40 Click **Save As** and then **OK** in the window that appears. This allows you to generate a full report.
41 The report looks like this. Take a closer look at the document. The report allows you to make an informed choice between the original and the alternative material selection.

42 If you click Online Info, additional information on the impact of your design is provided. This includes impact equivalents ‘such as the carbon footprint in miles driven in a hybrid car.

43 Setting a baseline allows you to compare the current selections against different selections.
   1 This button allows you to set and lock the Baseline.
   2 This button allows you to import the Baseline.

44 Close SustainabilityXpress.
   1 Now try a few other materials yourself to see which is the best solution (e.g. wood).
   2 Click on the red X.
Now, make a drawing for use in the workshop.

1. Click **New** in the toolbar.
2. Select the **Drawing** template.
3. Click on **OK**.

In the menu that appears, choose **B (ANSI) Landscape**.
Click **OK**.
Lesson 1: Base Sheet

47 An empty drawing field is displayed. The Model View command appears in the PropertyManager.

1 Make sure the appropriate part has been selected.
2 If not, use the browse button to find the part.
3 Click the arrow to continue.

48 Set the following options in the PropertyManager.

1 Select Create multiple views.
2 Choose the front, side, and isometric view in the Orientation menu.
3 For the Display Style, choose Hidden Lines Visible.
4 Leave the sheet scale at: Use sheet scale.
5 If all went well, SolidWorks has positioned the three chosen views. Click on OK.
49 Now select the isometric view. Click **Shaded with Edges**. Click on **OK**.

50 Distribute the views on the drawing as in the figure.

51 Change the units to MMGS.
Add a detailed view to the drawing.

1. Click **Detail View** in the **View Layout** tab.
2. Draw a circle as illustrated.

Enter the following information in the **PropertyManager**:

1. Type a capital letter A for the label of the detail.
2. We want to enlarge the detail. Therefore, choose: **Use custom scale** and select **User Defined** from the drop down.
3. Enter the ration 5:1.
4. Click next to the top view to position the detail.
54 Draw a **centerline** from the center of the circle straight up.

1. Click the arrow next to the line tool to show the **Centerline** function and select it.
2. Click in the middle of the circle; draw the **Centerline** and click anywhere above the model.

55 You will add two more lines using the function **Circular Sketch Pattern**.

1. Click the drop down under **Linear Sketch Pattern** and select **Circular Sketch Pattern**.
2. Enter 3 as a quantity.
3. Click in the **Entities to Pattern** window and select the vertical line you have just drawn.
Lesson 1: Base Sheet

56 Dimension the drawings as in the illustrated figure.
This completes your working drawing. Save the file as Base Sheet.SLDDRW.

57 List the most important things you have learned during this lesson.
Lesson 2: Windmill Assembly

In this exercise, you will get acquainted with assembled products: Assemblies. This assembly will consist of the part you made in the previous lesson, parts that you have downloaded, and parts that you get from the Toolbox. In this tutorial, you will learn how to connect one piece to another.

Work Plan

You will assemble a windmill. You will use the part that you have made yourself and the parts that you have downloaded.

- First, you will learn how to bring pieces into the Assembly environment.
- Next, you will learn how to assemble the pieces (mate).
- You will learn how to use the Toolbox.

Before you begin, make sure that you have downloaded and unzipped the necessary parts.
1 Launch SolidWorks.

1 In the menu bar, click New.

2 In the displayed menu select Assembly.

3 Click OK.

2 In the Begin Assembly PropertyManager, click Browse. Using this command, search for the appropriate part. We are looking for Housing.SLDprt.

If this PropertyManager does not automatically start, click Insert Components in CommandManager.
3 Go to the folder where you saved your models.

1 Select the model Housing.SLDPRT.

2 Next, click Open.

4 The part is now attached to the cursor. Click OK to place the part at the origin. Placing the part at the origin is very important for correct assembly of the entire product.

5 In the CommandManager, click Insert Components to add the next piece to the assembly.
We will begin searching for the next part.

1. Click **Browse**.
2. Next, select the part **Cap Internal.SLDPRT**.
3. Click **Open**.

Click anywhere in the drawing area to add the part. The added part is now positioned at an arbitrary location in the assembly.

Now, we will connect both parts together. In the **CommandManager** click **Mate**.
You must now select two elements between which a **Mate** will be made.
This needs to be done very carefully.
Zoom in on the top section of the housing.
1. Select the inner *edge* of the hole.
2. In the blue area of the **PropertyManager**, the edge will be displayed.

Rotate the model so the bottom of the **Cap Internal** becomes visible. To achieve this, press the scroll wheel of the mouse or use the arrows on the keyboard.
Select the edge of the **Cap Internal** as illustrated. Make sure you did not select a face.

**Tip:** You may accidentally choose the wrong face or edge. In that case, do the following:
With the right mouse button, click in the blue **Mate Selections** box.

Then click **Delete** to remove the selected part (displayed in dark blue in the window).
Click **Clear Selection** to remove everything.
If necessary, move the upper cap as illustrated. Select the top face of the hole.

Select the face on the bottom of the Cap Internal as illustrated.

Both parts now move to each other.

1. In the blue area of the PropertyManager, the following is displayed: Face<1>Housing-1 Face<2>Cap Internal-1

2. For the Mate, SolidWorks has selected Coincident this time.

3. Click OK to confirm the Mate.

The selection area in the PropertyManager is emptied, so you can immediately enter the next mate.

To fasten the cap, we mate the Front Plane of each part. However, it cannot be selected in the model but only in the FeatureManager.

Since the PropertyManager and the not the FeatureManager is displayed, you have to use the FeatureManager displayed in the graphics area.

Click the plus sign in front of the file name.
15 Click the plus signs for both parts. **Attention!** After having clicked the first +, the list scrolls.

16 Add the last mate between the **Housing** and the **Cap Internal**.

1. Within the **Housing** part, select the **Front Plane**.
2. Within the **Cap Internal**, also select the **Front Plane**.
3. The selected planes are displayed in the blue area of the **PropertyManager**.
4. SolidWorks has selected **Coincident** as the **Mate**.
5. Click **OK** to confirm the **Mate**.
6. Click **OK** once more to close the **PropertyManager**.

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17 Now, add the other required parts.

You can do that by repeating steps 5 through 7.

Add the following parts:
1  Wing Arm
2  Shaft
3  Base Sheet
4  Housing Base
5  Phone Holder
6  Windblade
7  Bottom End
8  Magnet Holder
9  Top End

18 We will now continue with the windmill assembly.

1  Drag the Base Sheet and the Housing Base somewhat downward.
2  Click the arrow below Move Component to open the scroll down menu.
3  Select Rotate Component.

19 Rotate the Housing Base as show in the figure.
Click OK.
20 Zoom in at on the **Base Sheet** to see on which side the recessed holes are located. If visible, rotate the part in such a manner that the recessed holes will be on the bottom side.

![Zoom in at on the Base Sheet to see on which side the recessed holes are located.](image)

21 Mate the two parts.

1. Click **Mate**.
2. Zoom in on the **Housing Base** and select the lower edge of the hole.
3. For the **Base Sheet**, select the upper edge of the hole.
22 Once you have clicked the upper edge of the hole, the parts start moving towards each other.

1 The selected parts are displayed in the blue area of the PropertyManager.

2 SolidWorks selected Coincident as the Mate.

3 Click OK.

23 Add the last mate between the Housing Base and the Base Sheet.

1 First, select the lower edge of the Housing Base. Then select the upper edge of the hole in the Base Sheet.

2 Coincident is the appropriate mate, so we’ll leave that selected.

3 Click OK.

4 Click OK once more to exit the function.

24 Drag the Bottom End toward the Housing as shown in the figure.
25 Mate the **Housing** and the **Bottom End** as is steps 8 through 13.

1 Click **Mate**.

2 Within the part **Housing**, select the **Right Plane**.

3 Within the part **Bottom End**, select the **Front Plane**.

If necessary, click the plus sign to open the list of both parts.

The **Front Plane** of the **Bottom End** now turns towards the **Right Plane** of the **Housing**.

4 SolidWorks has selected **Coincident** as the **Mate**.

5 The keyway must be on the right side. If not, read the tip below.

6 Click **OK** twice.

**Tip:** To get the keyway on the right side, use the **Flip Mate Alignment** option.

26 Zoom in, drag and/or rotate the **Housing Base** and the **Base Sheet** as illustrated.

Both are linked together, which is why they move together.
27 First, **Mate** both keyways together.
   1. Click **Mate**.
   2. Click the upper edge of the **Housing Base**.
   3. Next, select the lower edge of the **Bottom End**.
   4. Create the mate by clicking **OK**.

28 Add the last mate between the **Housing Base** and the **Bottom End**.
   1. Select the upper edge of the hole of the **Housing Base**.
   2. Click the lower edge of the vent hole of the **Bottom End**.
   3. Click **OK** once.
   4. Finally, click **OK** to close the command.
29 Zoom in, drag and/or rotate the **Shaft** and the **Magnet Holder** as illustrated.

1. Next click on **Mate**.
2. Select the outer face of the **Shaft**.
3. Then choose the inner face of the **Magnet Holder**.
4. Using a **Concentric** mate, link both parts together.
5. Click **OK**.

30 Add a distance mate.

1. Select the upper face of the **Shaft**.
2. Then choose the upper face of the **Magnet Holder**.
3. For mate, select **Distance**.
4. For the distance, enter **150 mm**.
5. Check or uncheck **Flip dimension** to move the **Magnet Holder** to the right place.
6. Click **OK**.
31 Mate the Front Plane of the Shaft and the Magnet Holder.

If you don’t know how to do that review steps 14 through 16 or step 25.

32 To finalize this piece, you must position two more parts.

You need a Magnet and an M6x8 Allen head set screw.

1 Click Insert Components.

2 Go to the folder where you saved the parts.

3 Add the magnet.

33 Mate the outer face of the Magnet with inner side of the hole.

Then click OK.
Add the final mate between the Magnet and the Magnet Holder.

1. First, select the outer face of the Magnet Holder.
2. Then select the top face of the Magnet.
3. In this case, SolidWorks has selected a Tangent mate.
4. Click OK to confirm the mate.
5. Click OK once more.

We will now add a M6x8 mm Allen head set screw. We will do that using the SolidWorks Toolbox function. Before continuing, you must first make sure the Toolbox has been installed and activated on your computer.

1. In the CommandManager click the arrow next to the Options icon.
2. Select Add-Ins.

Make sure the SolidWorks Toolbox and SolidWorks Toolbox Browser options are both checked in the menu.

By adding a check mark to the right of both options (SolidWorks Toolbox and SolidWorks Toolbox Browser), they will be, from now on, automatically loaded when SolidWorks is launched. Therefore, you don’t have to activate the Toolbox each and every time.
37 In the **Task Pane** (on the right side of the screen), click the **Design Library** icon.

38 The **Task Pane** will open with the **ToolBox**. We will now insert an Allen head set screw into the threaded hole.

   Successively click:
   1. **Toolbox**
   2. **Iso**
   3. **Bolts and Screws**
   4. **Set Screws - Socket**
   The available screws will be displayed in the lower part of the **Task Pane**.
   5. Search for the following screw: **Socket Set Screw Flat Point ISO 4026**.

39 With the left mouse button, drag the screw from the **Task Pane** to your model. As soon as the mouse moves above the threaded hole, the screw jumps to the appropriate position. Release the mouse button.

   The screw may seem too small or too large. That is not important at this point.

   In the **PropertyManager**, change the size of the screw to **M6x8**, and click **OK**.
40 The screw is now locked to the mouse and you could insert it into other threaded holes.

But we don’t have any other holes so we no longer need the screw. Therefore, click **Cancel**.

41 Add a **Tangent** mate between the bottom of the set screw and the **Shaft**.

42 Zoom in, drag and/or rotate the **Housing Base** and **Base Sheet** as illustrated.
Add the following part in the same way as above. 

Successively click:

1  Toolbox
2  ISO
3  Bolts and Screws
4  Cross-recessed Head Screws

The available screws will be displayed in the lower part of the Task Pane.

5  Search for the following screw:  
   CTSK Flat ISO 7046-1.

Drag this screw to the hole. 

In the PropertyManager, change the size of the screw to M3x8, and click OK.
45 The screw is locked to the cursor so you can insert it into other holes as well. Add two more screws, then click **Cancel**.

46 The screws are still protruding from the base. Solve that problem by doing the following.

1. Click **Mate**.
2. Select the top face of the screw.
3. Select the face of the **Base Sheet**.
4. Click **OK**.
5. Repeat this for both other screws.
6. Finally click **OK** once more.

47 Add the follow part to the bolt ends in the same manner: Hex Nut Style 1 ISO - 4032 (M3 Bolt).
We will add a bearing. You will use the **Toolbox** once more.

Click the following:
1. Toolbox
2. ISO
3. Bearings
4. Ball Bearings
5. Drag **Angular Contact Ball Bearing** to the hole of the **Cap Internal**.

Insert the bearing into the hole.
Locate the appropriate bearing: **8315**.
Click **OK**.

Press **Cancel** to close.
51 Mate the **bearing** as illustrated.

52 Mate **Plane 1** of the **bearing** and the **Right Plane** of the **Cap Internal**. This is required to fasten the bearing.

**Tip:** Sometimes a part is in the way during the assembly. For instance, it may not be possible to select a part correctly. This can be solved in two ways.

You can hide a part that is in the way by clicking it and then selecting **Hide Components**.

You can bring it back again by clicking the hidden part in the **FeatureManager** and then selecting **Show Components**.

Or you can make the part transparent. Again, click the part and then select **Change Transparency**.

If you want the part to be displayed normally again, click it once more and then click **Change Transparency**.
Lesson 2: Windmill Assembly

53 Add the **Dynamo** to the assembly.

1. Click **Insert Components**.
2. Click **Browse**.
3. Change the search filter to **Assembly**.
4. Then choose the file **Dynamo.SLDASM**.
5. Click **Open**.

54 Click **Mate**.

Choose the outer edge of the shaft.
Then choose the interior of the **Bottom End**.
Click **OK**.
Click **OK** once more.
55 Select the bottom of the hole of the **Bottom End**.

56 Finish the distance mate.
   1 Select the bottom of the hole of the **Dynamo**.
   2 Select the **Distance** mate.
   3 Enter **4.17 mm**.
   4 Click **OK**.
57 Select the faces as shown in the figure.
Select a **Parallel** mate.
Click **OK**.
Click **OK** once more.

58 Now **Mate** the shaft of the dynamo to the hole of the drive shaft. See the figure.
59 Create another mate between the bottom of the **Shaft** and the top of the **Dynamo**.

60 Now it is time to put the **Top End** in place. Click **Mate** and select the upper face of the **Cap Internal** and the inner face of the **Top End**.

Mate these two parts together.
61 Select the threading of the shaft and the threaded M8 screw hole of the cap.

62 Click OK once.
63 Select the **Front Plane** of the **Shaft** and the **Front Plane** of the **Top End**.  
Click **OK**.  
Click **OK** once more.

64 Mount the **Wing Arm** to the **Top End**.  
Use the following mates to do this.  
**Concentric** for the shaft/ hole assembly.  
**Coincident** for the end shaft/ end hole mount.  
Finally, use planes to straighten the attachment rod. See the figure.
65 Now you can mount the Windblade.
First make a connection between the holes (Concentric). Next, link the outside of the Windblade to the Wing Arm (Tangent).

1 Finally, choose the Front Plane of the Wing Arm and the Top Plane of the Windblade.
2 Use a Perpendicular Mate.

66 Get a rivet from the Toolbox.
Click the following:
1 Toolbox
2 GB
3 Rivets and Studs
4 Choose Rivets.
5 Drag Flat round head drive rivet GB/T15855.1-1995 to the hole of the Wing Arm.
This should automatically create a mate between the rivet and the Wing Arm.
67 Set the following:
2. Rivet length: 10.
3. Click OK. In the next screen click Cancel.

68 For the following features we need a guideline running through the middle of the model. This axis already exists in the model, but is invisible (in the standard settings).
1. Click Hide/Show Items.
2. Make sure the button View Temporary Axes is activated.

69 Choose the function: Circular Component Pattern.
Lesson 2: Windmill Assembly

Set the following:

1. Choose the axis of cap.
2. The window displays which axis you have selected.
3. Enter 360°.
4. In the PropertyManager change the number of copies to 4.
5. Select the Windblade, the Wing Arm, and the rivet.
6. Click OK.

Insert and Mate the Phone Holder as shown in the figure.

Add the Phone Holder once more.
In the FeatureManager click the second instance of the Phone Holder.
Click Component Properties.
73 A new menu is displayed.

1. Choose **Digital Speedometer**.
2. The choose **OK**.

**Tip:** Another way to change the configuration of a part is simply to click on the part, and select the configuration from the drop down. Finally, click on **OK**.
74 Mate it in the same way as the **Phone Holder**. See the figure.

75 Insert the phone and the speed indicator. Next, mate the phone and the speed indicator to their holders.
Finally, add the **Magnet Detector** to the assembly. Then, mate the **Magnet Detector**, the **Housing**, and the **Bottom End**.

When the assembly is ready, save the file as **Windmill.SLDASM**.

List the most important things you have learned during this lesson.