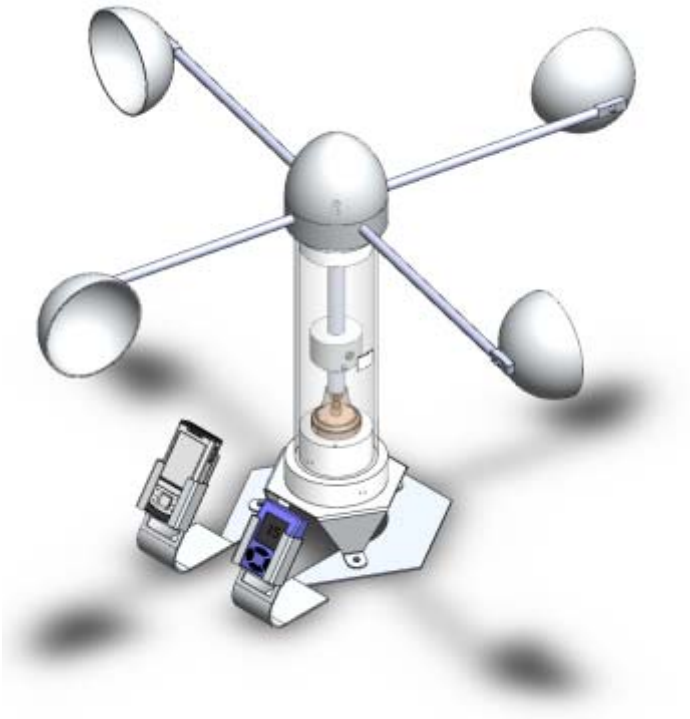
SolidWorks® tutorial 13-5
“Exciting”

Includes SolidWorks SustainabilityXpress

“A better world for our children and grandchildren”



Pre-vocational Secondary Education
and Senior Secondary Vocational Education

Stichting Consortium Beroepsonderwijs

SolidWorks

platform metaal & metalektro

For SolidWorks® Educational Release 2009-2010

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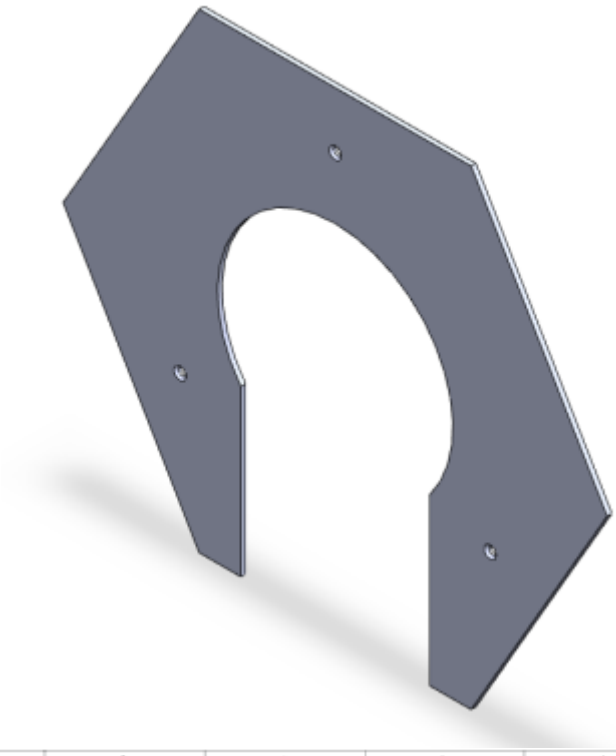
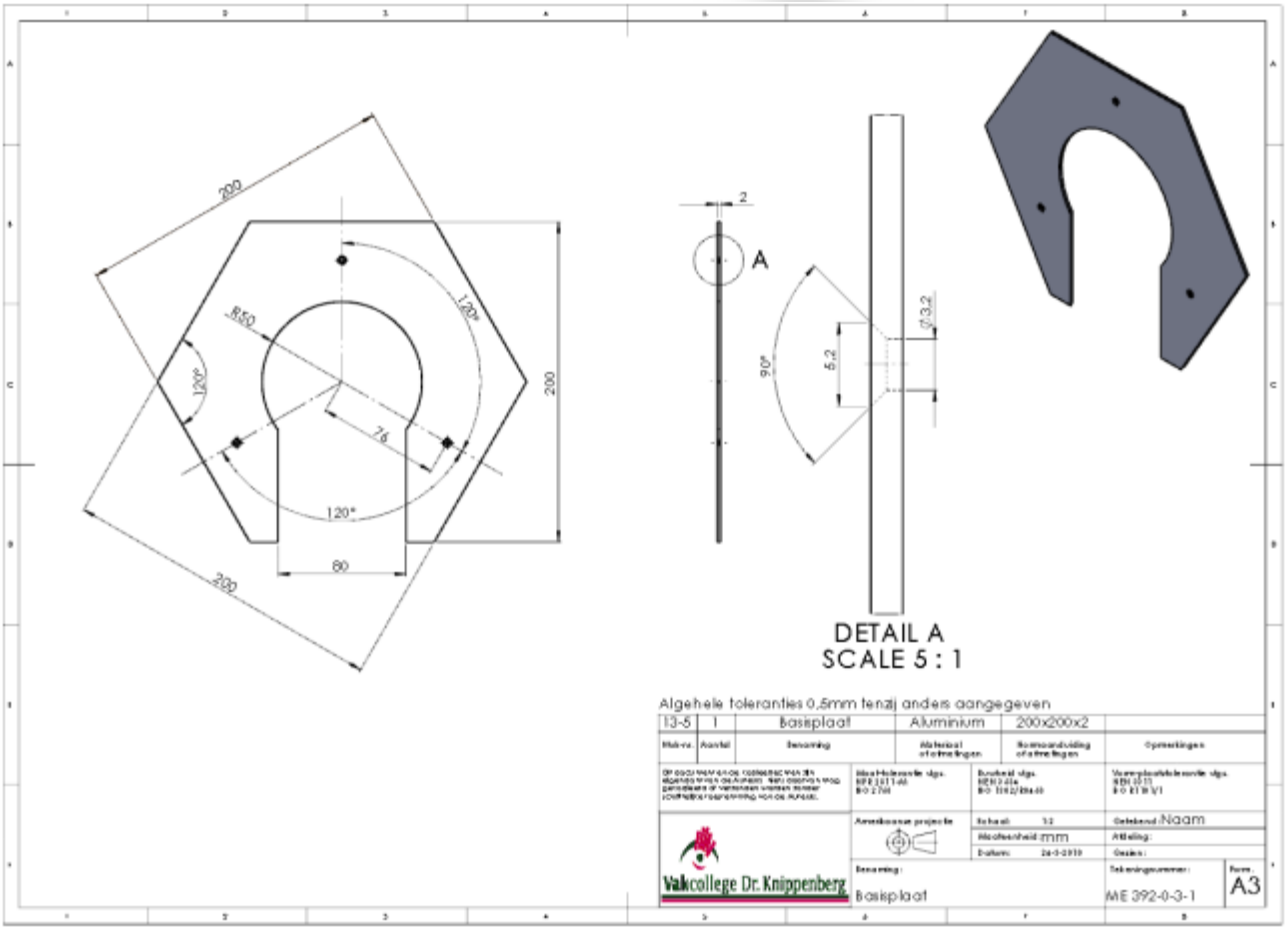
Adaptation to the educational level: Jack van den Broek (Technical school Dr. Knippenberg).
Completed by: Nenad Raskovic

Tutorial 13-5
“Base Sheet”



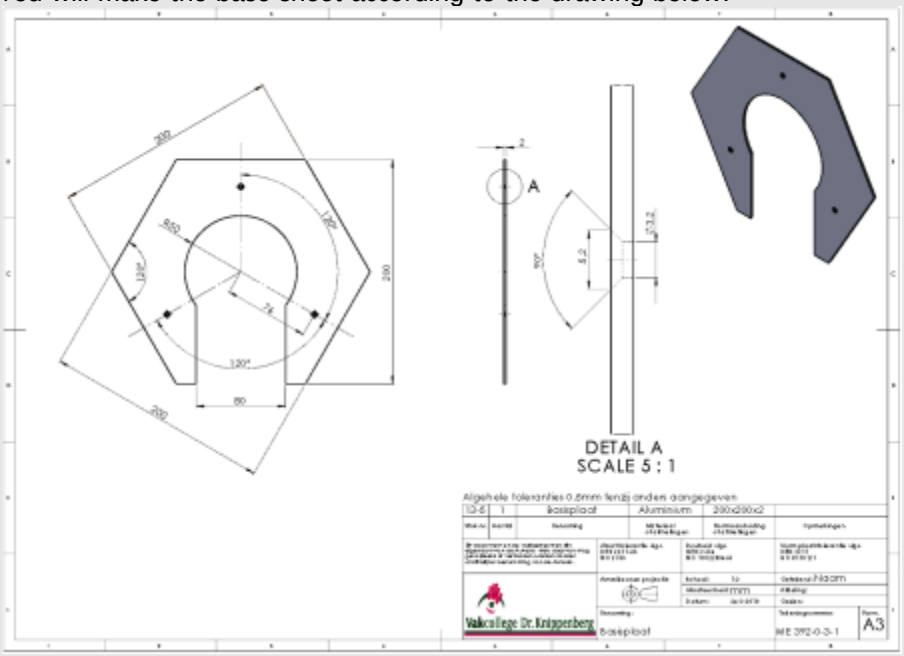


Base Sheet

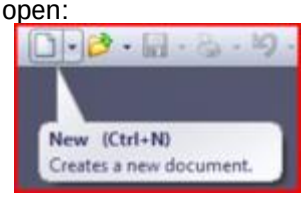
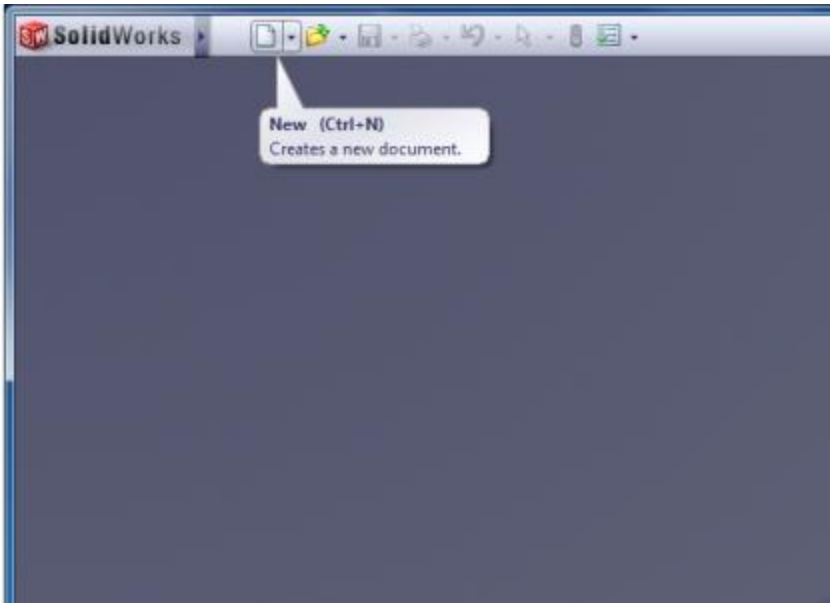
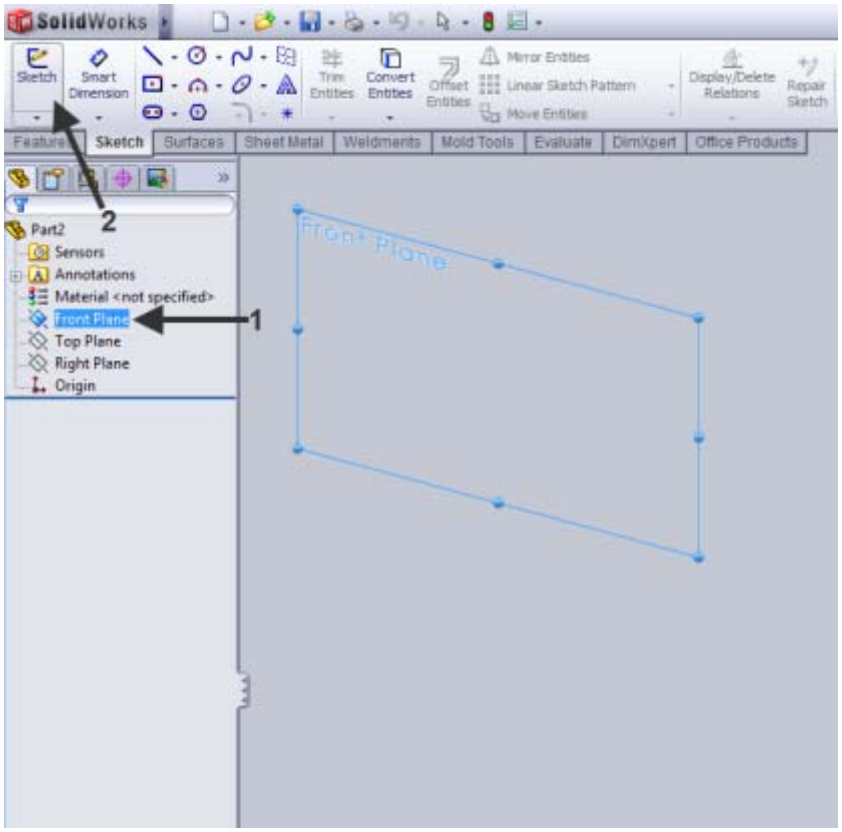
In this exercise, we will make a simple aluminum sheet.
You will get acquainted with Sheet Metal as a function in SolidWorks.
And, of course, you will make a working drawing of this piece.
The Tutorial will also present **SolidWorks Sustainability Xpress.**
We do this because we want to protect the environment.

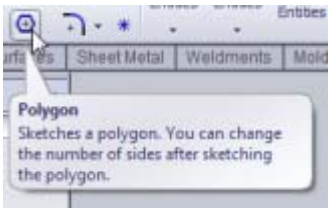
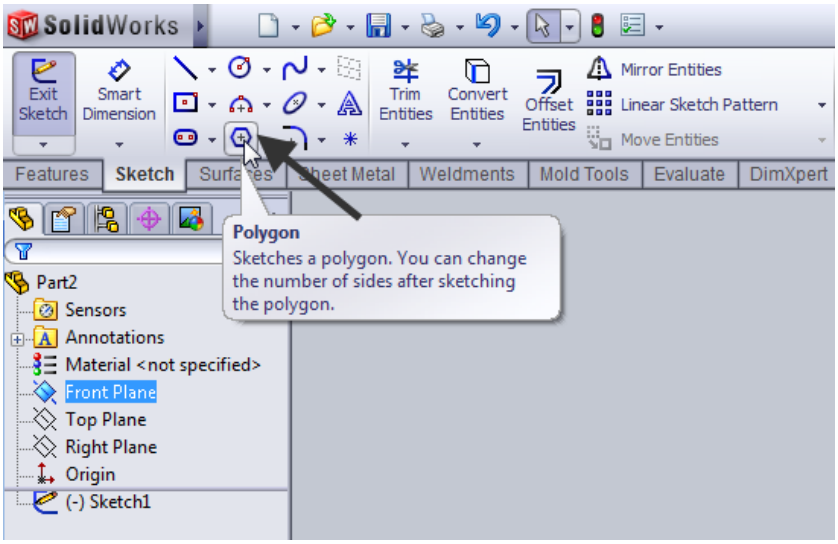
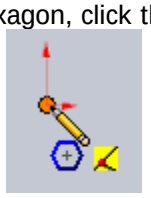
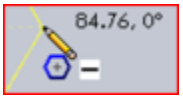
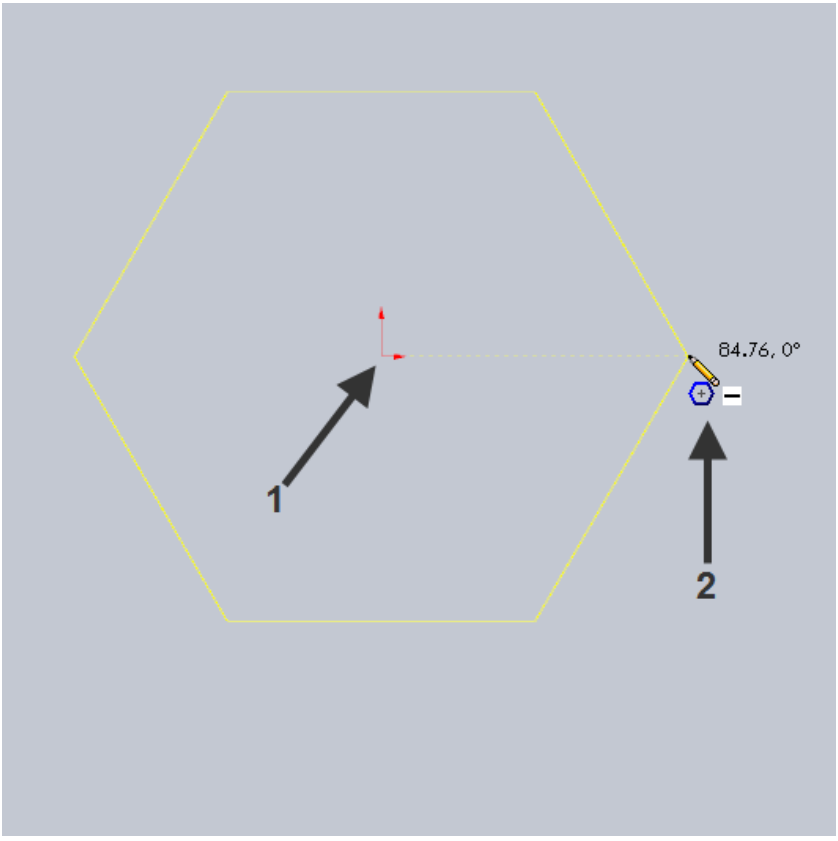
Work plan:

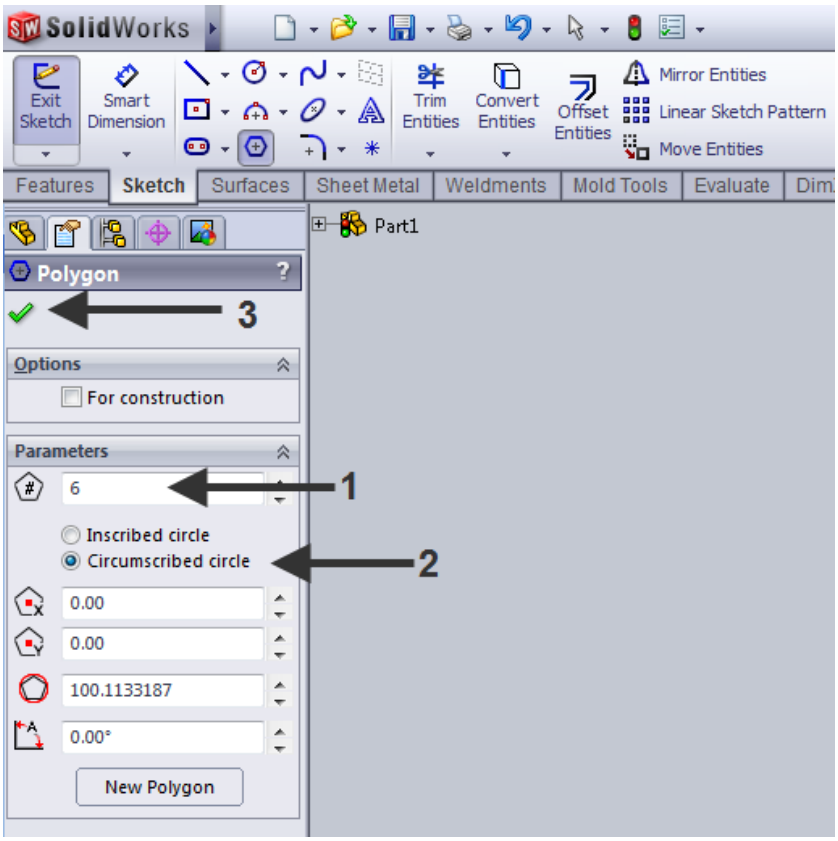
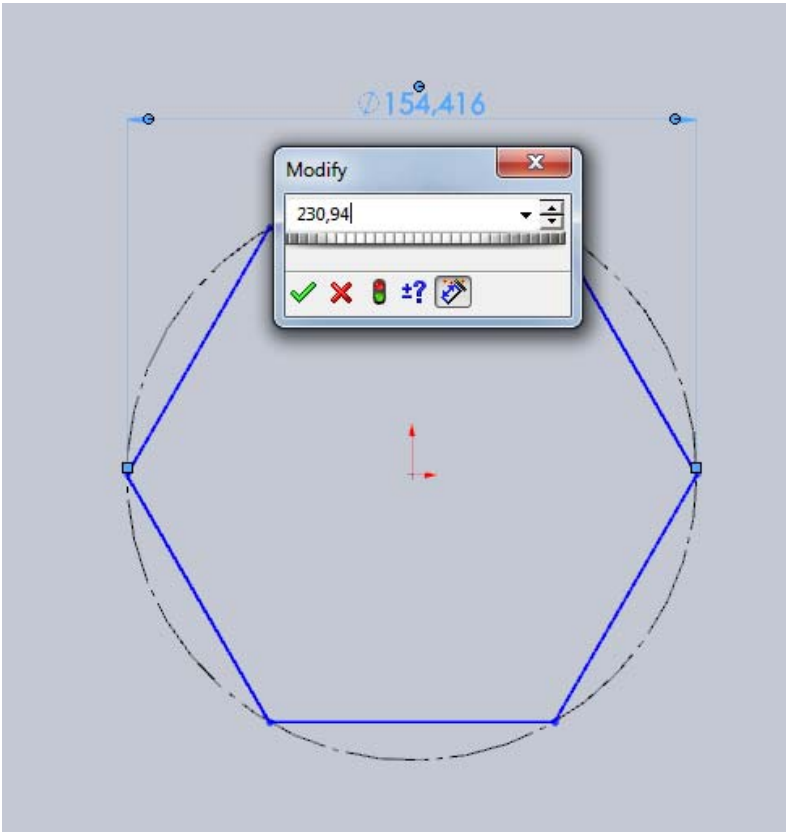
You will make the base sheet according to the drawing below.

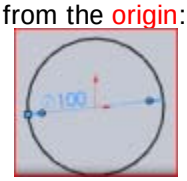
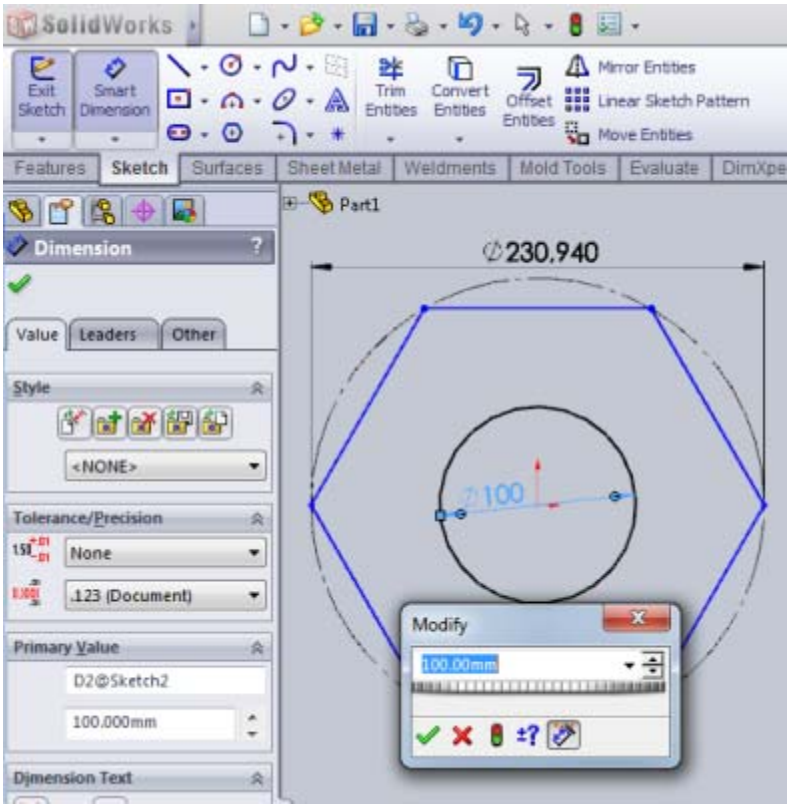
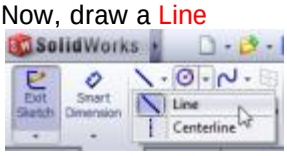
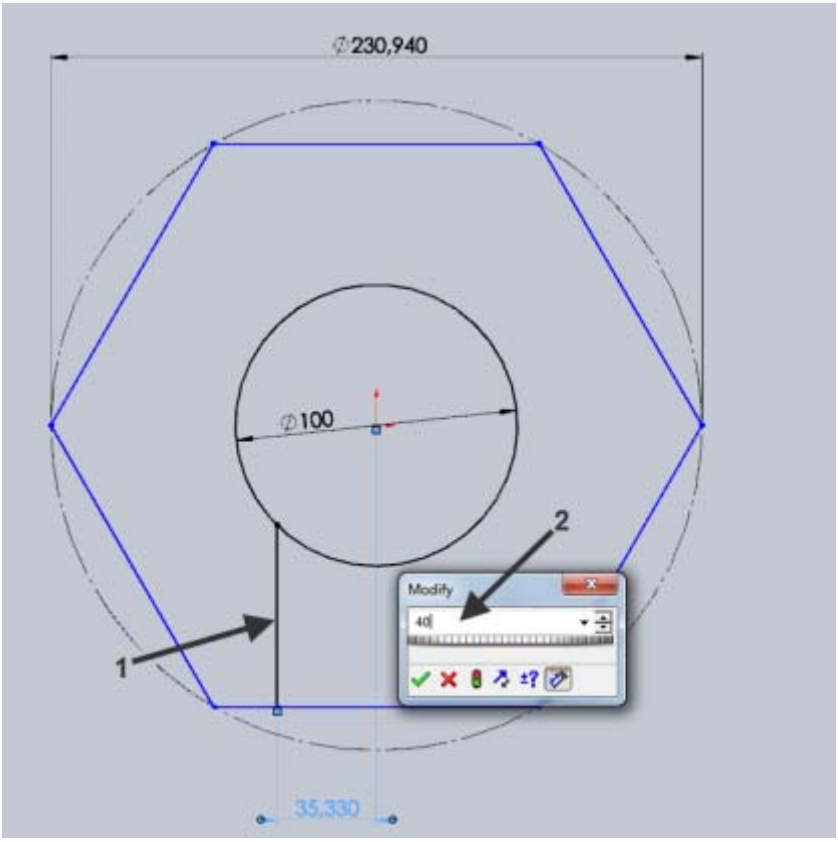


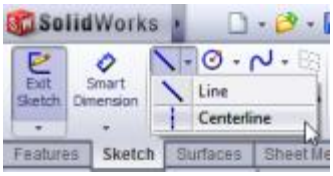
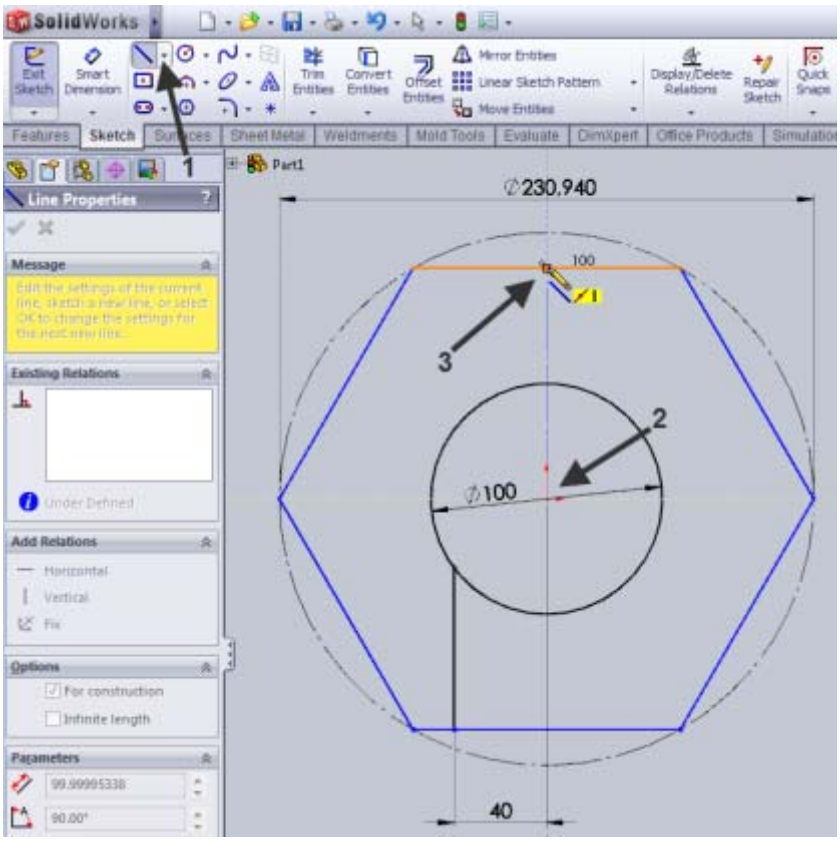
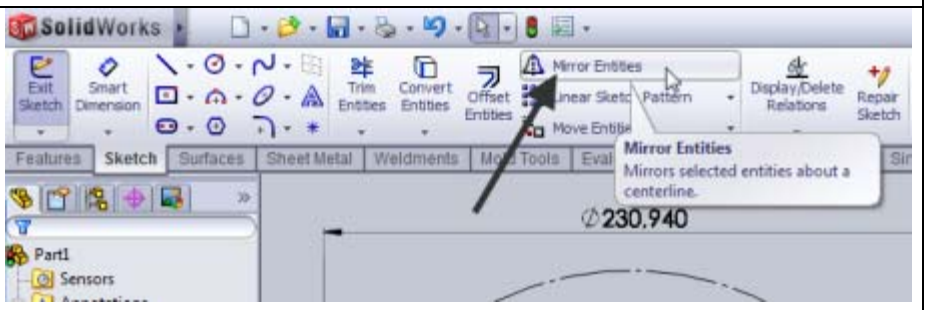
1. First, draw a hexagon. (Polygon)
2. Sketch a circle starting from the center of the hexagon.
3. Next, make an opening at the bottom of the circle and the hexagon.
4. After this, draw a pattern of 3 holes.
5. Finally, make a working drawing for use in the workshop.

1	Launch SolidWorks and open:  You do this to create a new document.	
2	**1.** Select:  **2.** Next, click on:  You do this to activate the Sketch environment.	

3	The base sheet is a regular hexagon. Click in the: Command-Manager on Polygon.  In this exercise, we will draw a regular hexagon.	
4	Draw a hexagon from point zero. **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 to work horizontally! See icon: 	

5	Make sure that in the: Property Manager, under: **Parameters** **1.** The number of sides is set to 6 6 **2.** A circumscribed circle now determines the size. Circumscribed circle **3.** Click OK.	
6	The size of the circle should become Ø 230.94mm. Because of the dimensioning of Ø 230.94mm, the size on the sides of the hexagon will be 200 mm. If you select the option: Inscribed circle Then the size displayed for the circle should be 200mm. Dimension it with the Smart Dimension function.	

7	Draw a Ø 100mm circle from the origin:  Next, dimension the circle.	
8	**1.** Now, draw a Line  straight to the top, as in the figure **2.** Then, dimension this line as in the adjoining figure. The distance between the line and the center of the hexagon is 40 mm.	

9	Draw a Centerline from the origin straight to the top. **1.** Click the function: Centerline:  **2.** For the first point, click the origin. **3.** Then, draw a Centerline straight to the top, as in the figure.	
10	We will now mirror the line. Select: Mirror Entities 	

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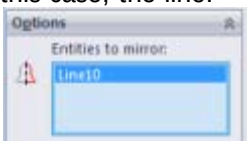
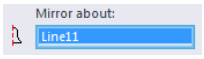
The Property Manager menu under Options shows the selected components you want to mirror.

1. First, select the line.
2. The window Entities To mirror displays the selected component, in this case, the line.

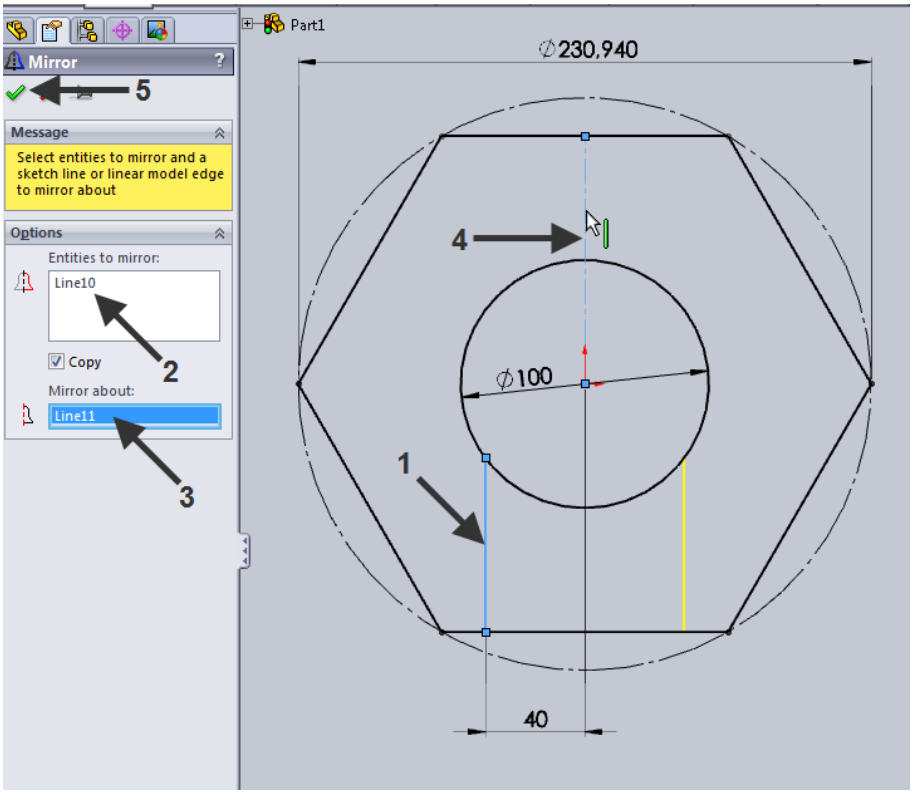


3. Click in the window:

4. Select the Centerline. Once you have selected the Centerline, it is displayed in the window Mirror about.



5. Click OK. 



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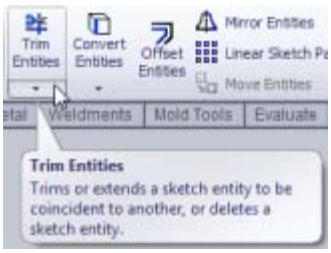
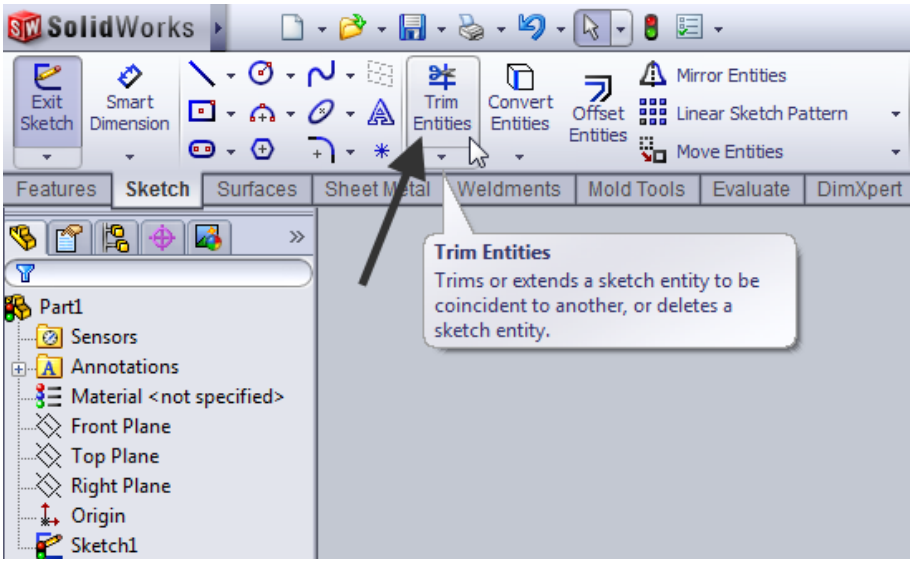
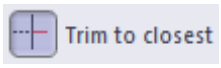
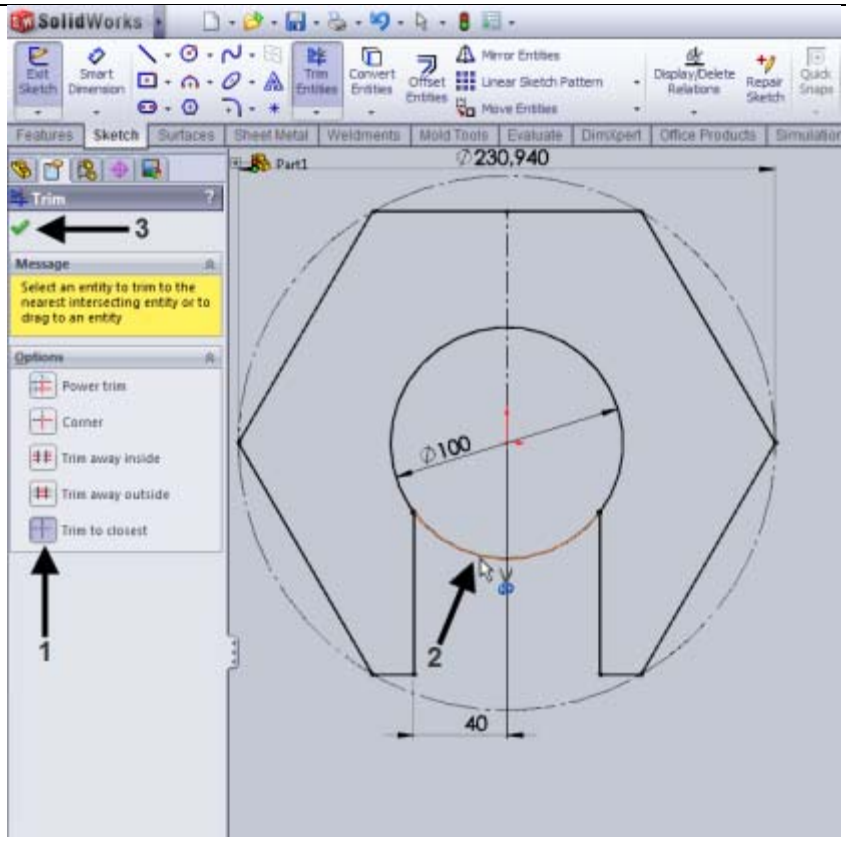
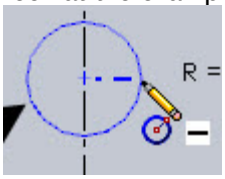
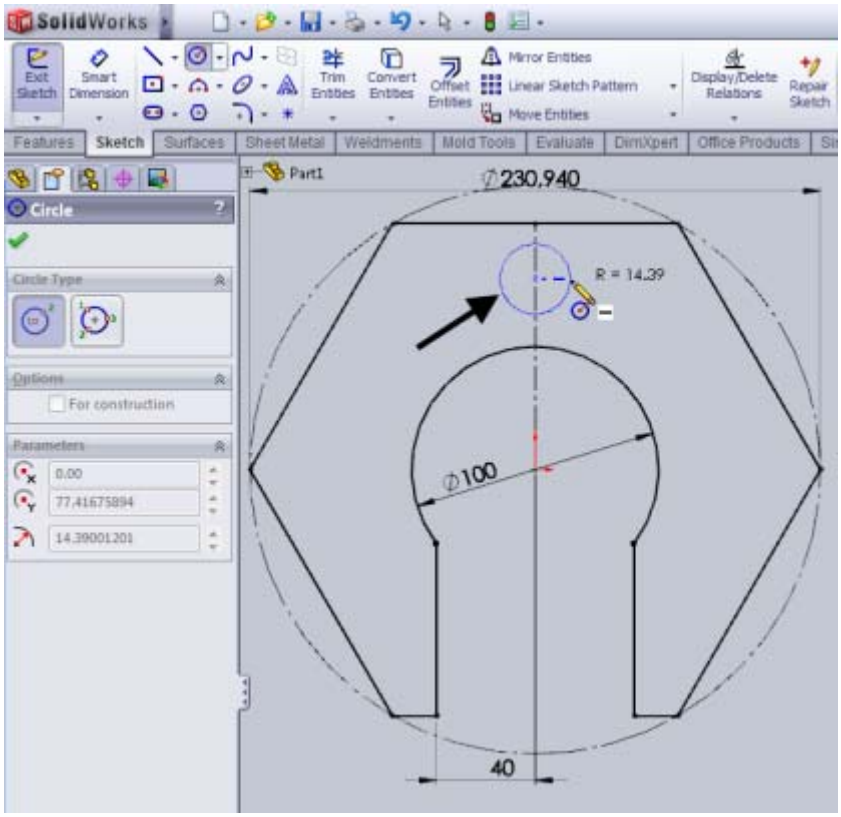
We want to remove the bottom of the circle and the hexagon to make a recess there.

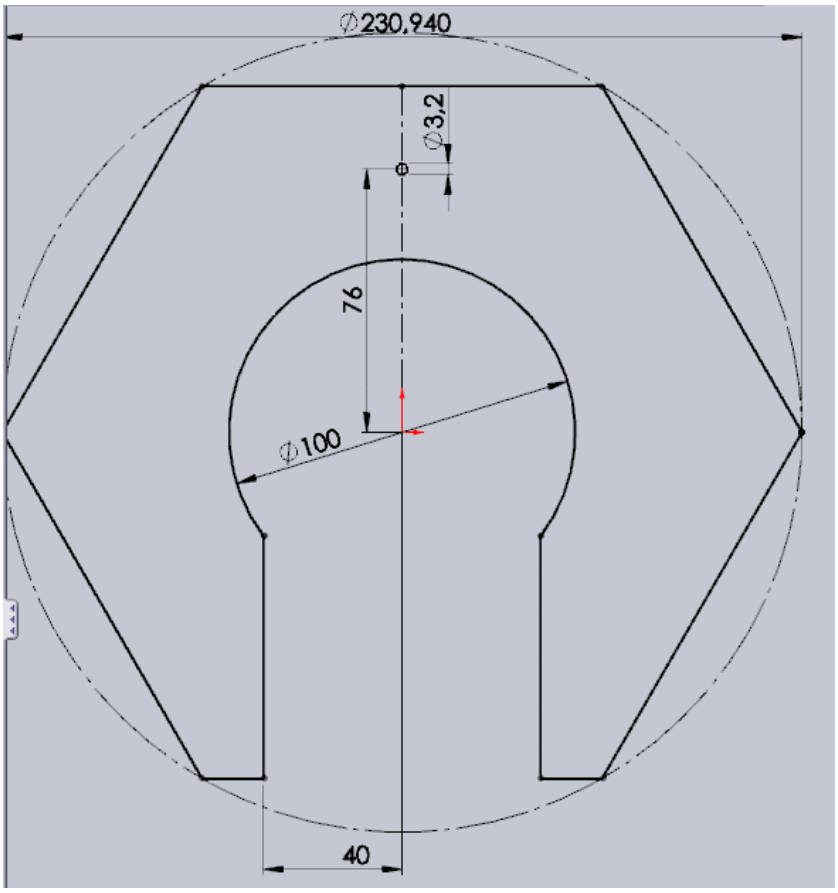
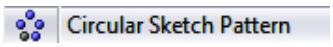
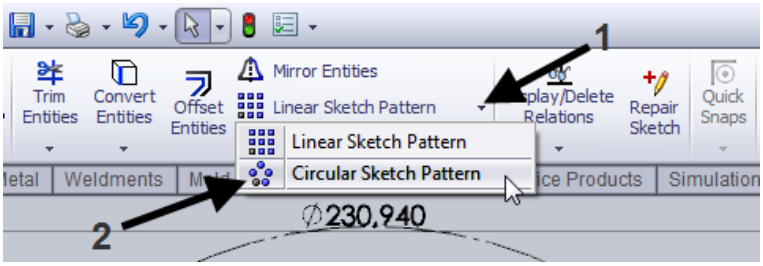
The bottom of the circle and the hexagon can be removed with the Trim Entities function.

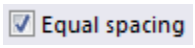
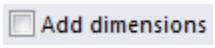
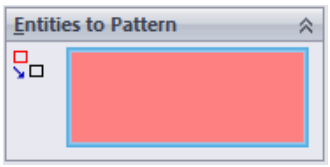
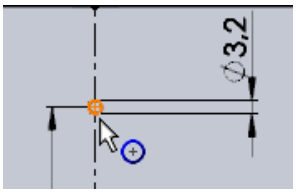
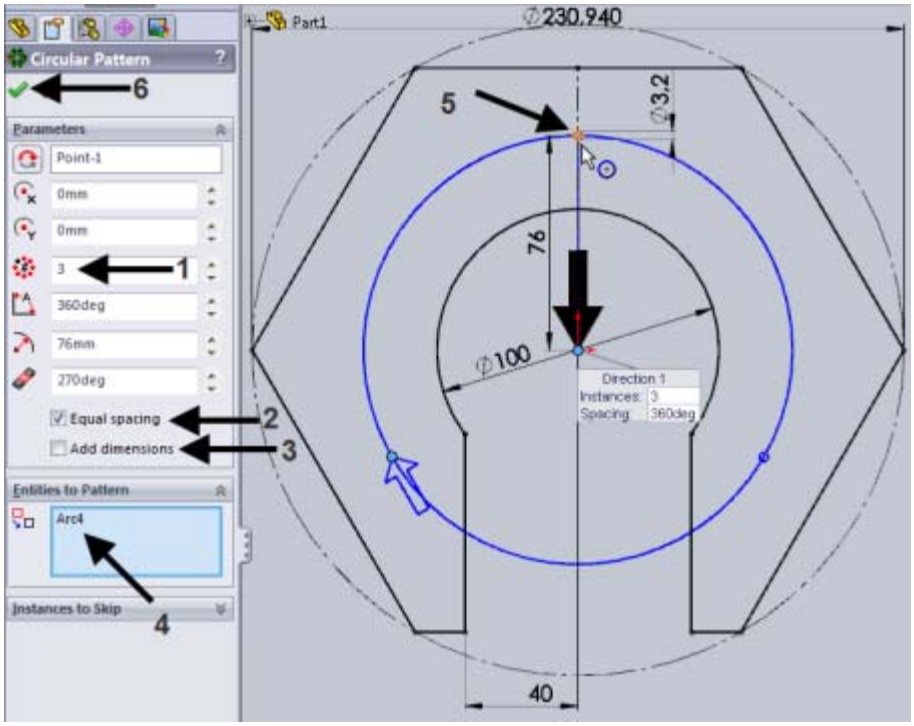


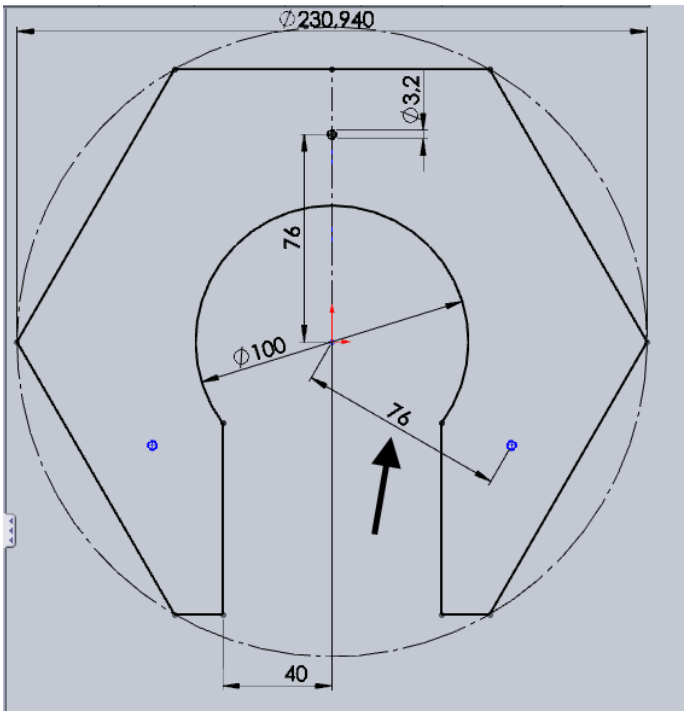
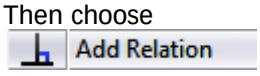
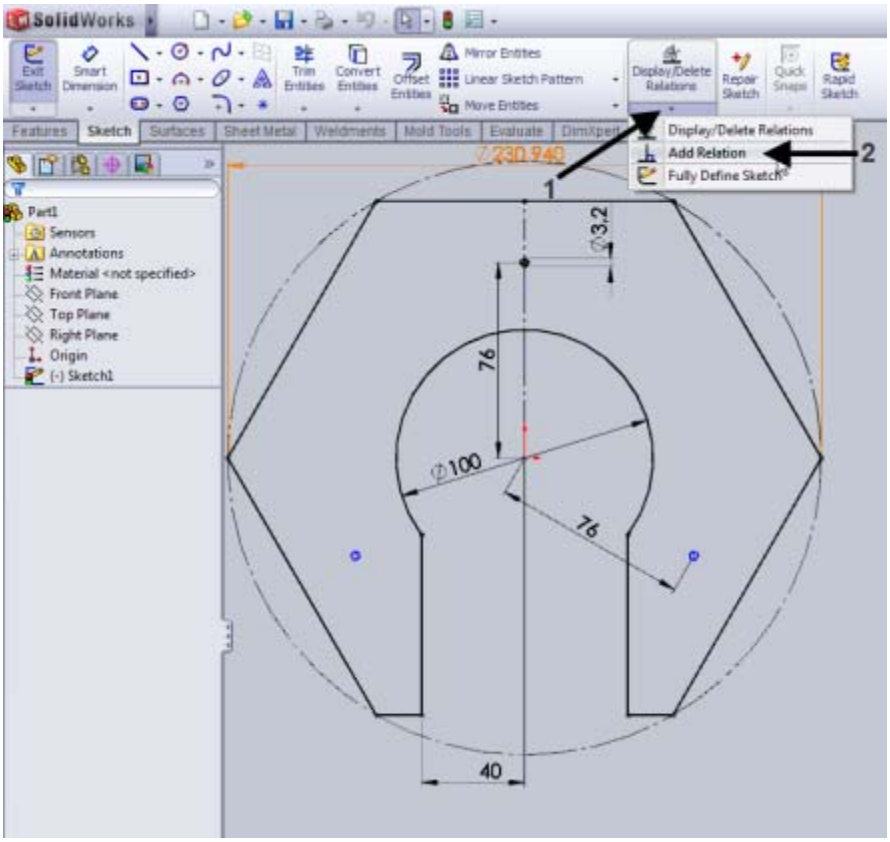
In the Command Manager, click Trim Entities

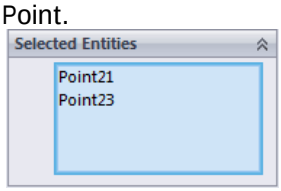
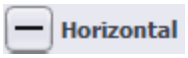
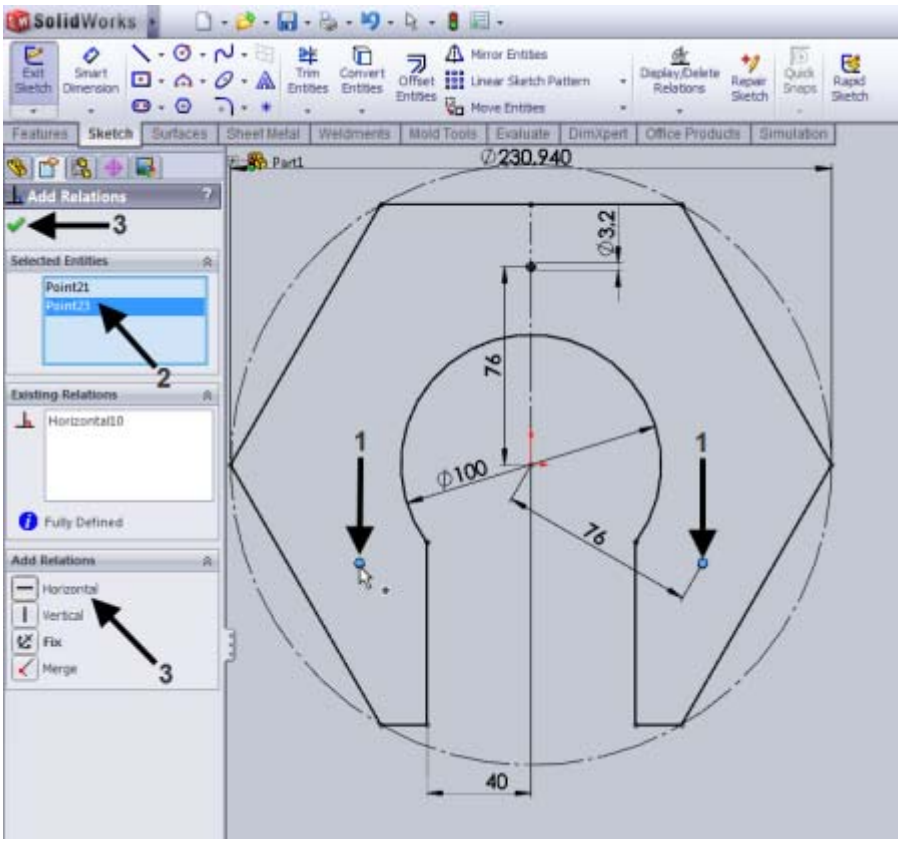
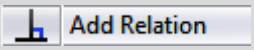
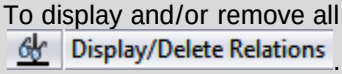


13	**1.** In the: Property Manager, select the option Trim to closest  **2.** Now, cut off the bottom part from the hexagon and the circle. In the example, the bottom part of the hexagon has already been cut off. **3.** Click OK.	
14	Now, draw a circle. The center of the circle must be on the centerline. Look at the example.  	

15	Enter the circle dimensions. The circle has a diameter of 3,2mm and a height of 76mm as measured from the origin.	
16	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. **2.** Select 	

17	Now, do the following: **1.** Set the number of copies to 3.  **2.** Check Equal Spacing.  **3.** Uncheck Add dimensions.  **4.** 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 displays the selected components. **5.** Select a 3.2mm circle.  **6.** Click OK.	
		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.

18	Using the Smart Dimensions function, dimension the circles you have just drawn. Also, enter 3.2mm as the dimension for both circles. The adjoining figure shows how this should look like.	
19	The sketch still hasn't 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 Display/Delete Relations  **2.** Then choose 	

20	**1.** Select the center of the two lower circles.  The plus sign shows you will be selecting the center. **2.** The selected objects will appear in the blue window, in this case, Point.  **3.** Then, click the Horizontal button  This will put both circles on one line.	
		The function  allows you to add various relations to the objects. 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  .

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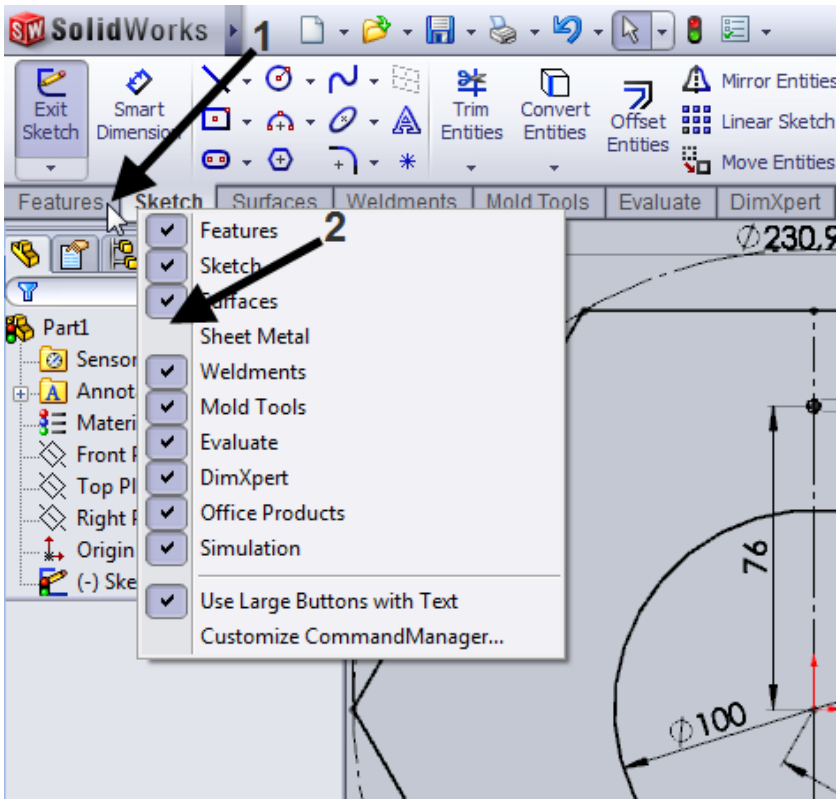
First, make sure the Sheet Metal buttons are available.

The best way to do this is to add them to the Command Manager.

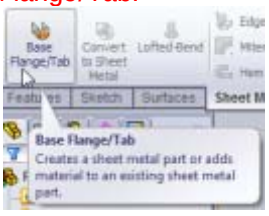
1. With the right mouse button, click a tab in Command Manager.
2. In the displayed menu, click: Sheet Metal.

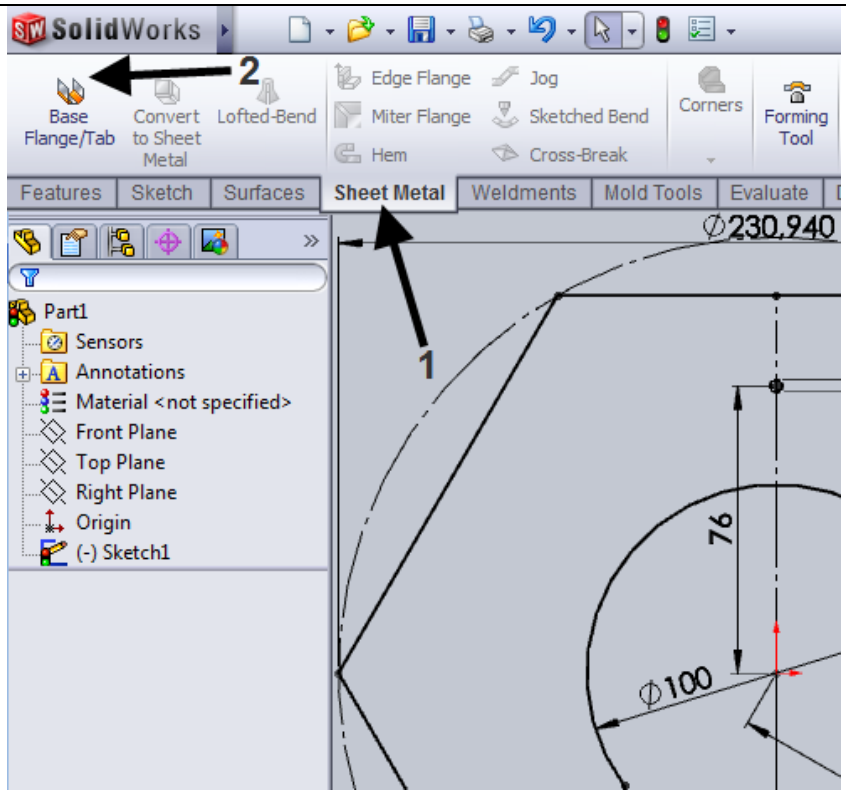
ATTENTION!!
If Sheet Metal is already checked in your system, continue to step 20.

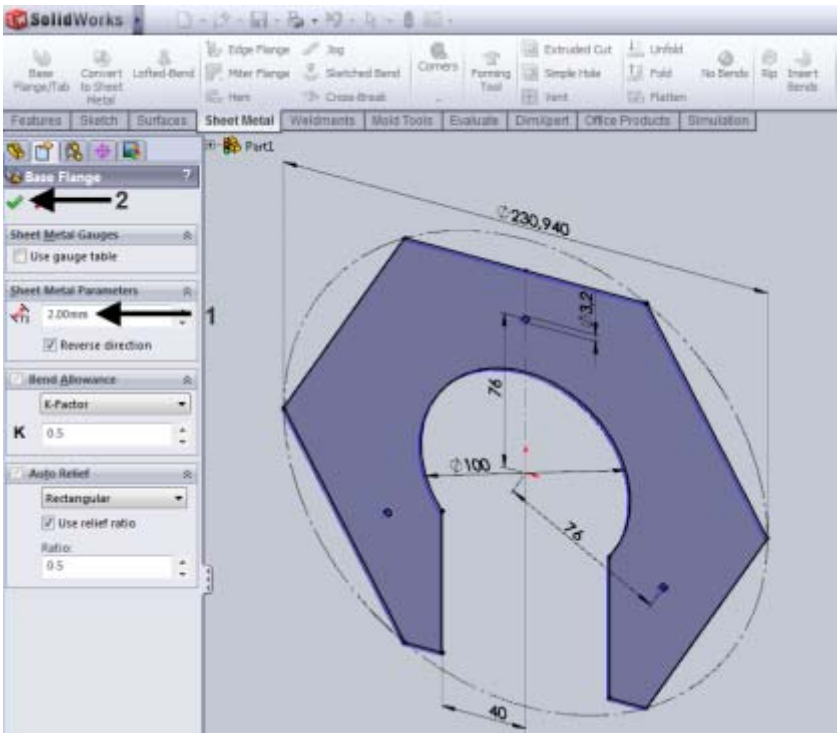
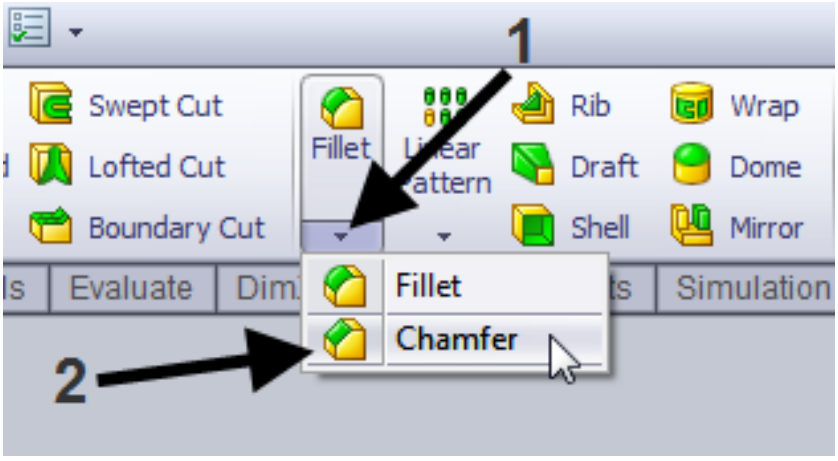


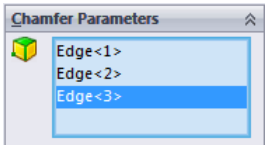
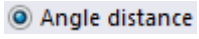
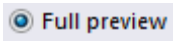
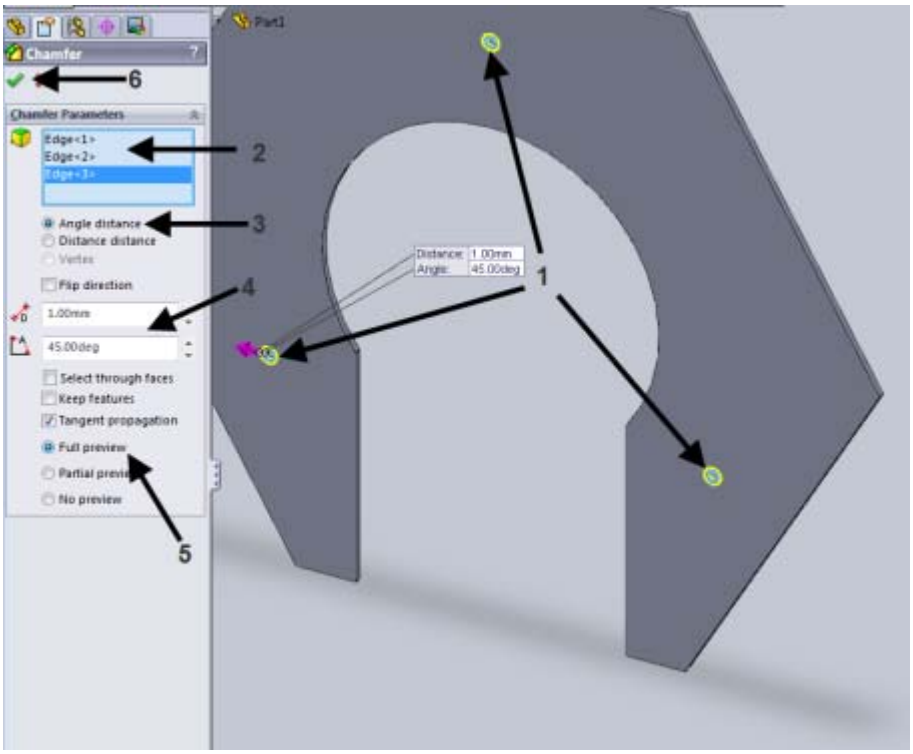
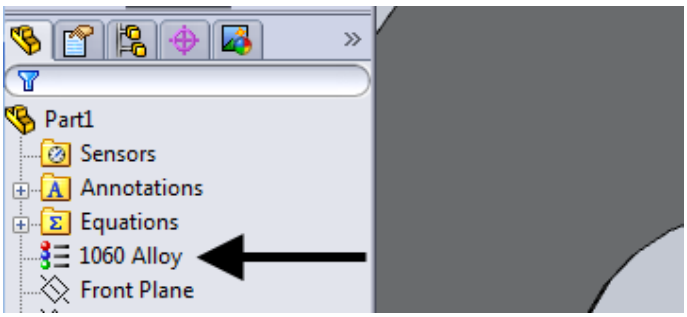
22

1. In: Command Manager, first click: Sheet Metal.
2. Then, click Base Flange/Tab.





23	**1.** In: Property Manager, enter 2mm as material gauge.  **2.** Click OK.  Leave the rest of the menu unchanged.	
24	Next, we will make the chamfer at the top of the circle. **1.** In the: Command Manager, click the arrow  under Fillet. **2.** Click Chamfer. 	

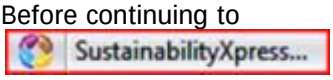
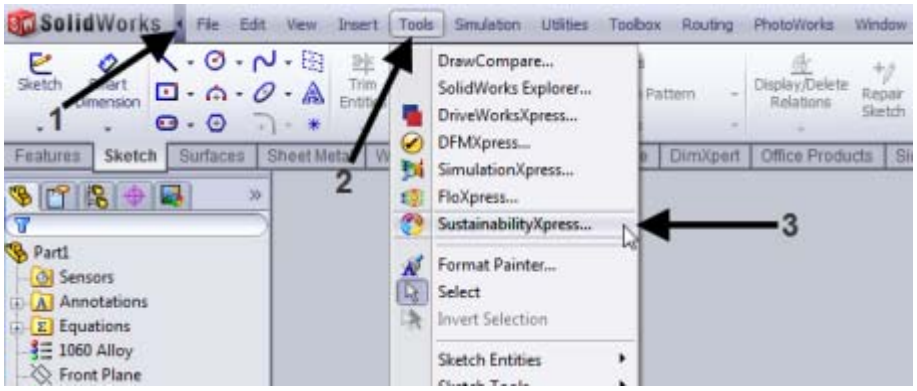
25	You must now set and verify a few things. **1.** Select the top edge of all three circles. (Use the CTRL key)  **2.** In the blue area, Edge has now been selected three times.  **3.** Make sure the 'Angle distance' option is selected. If not, check it.  **4.** As the chamfer distance, enter 1mm and 45 deg.  **5.** Make sure the option 'Full preview' is selected.  The model shows exactly what will happen. **6.** If everything has been set correctly, click OK. 	
26	Change the material to 1060 Alloy. The model is now ready. In the toolbar, click Save and name the file: **Base Sheet**	

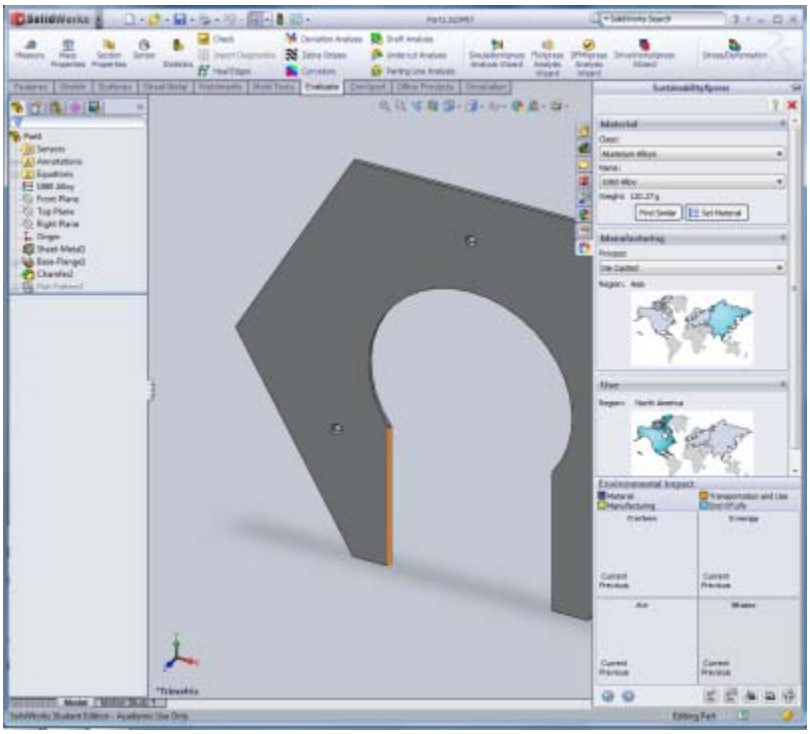
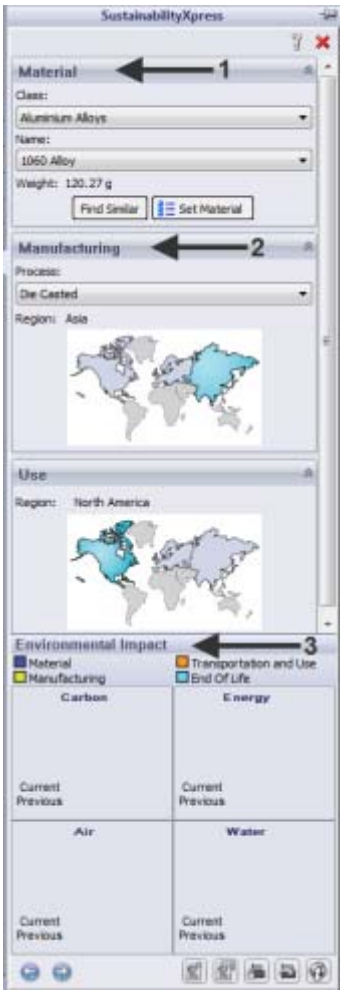
SolidWorks Sustainability Xpress

"A better world for our children and grandchildren"

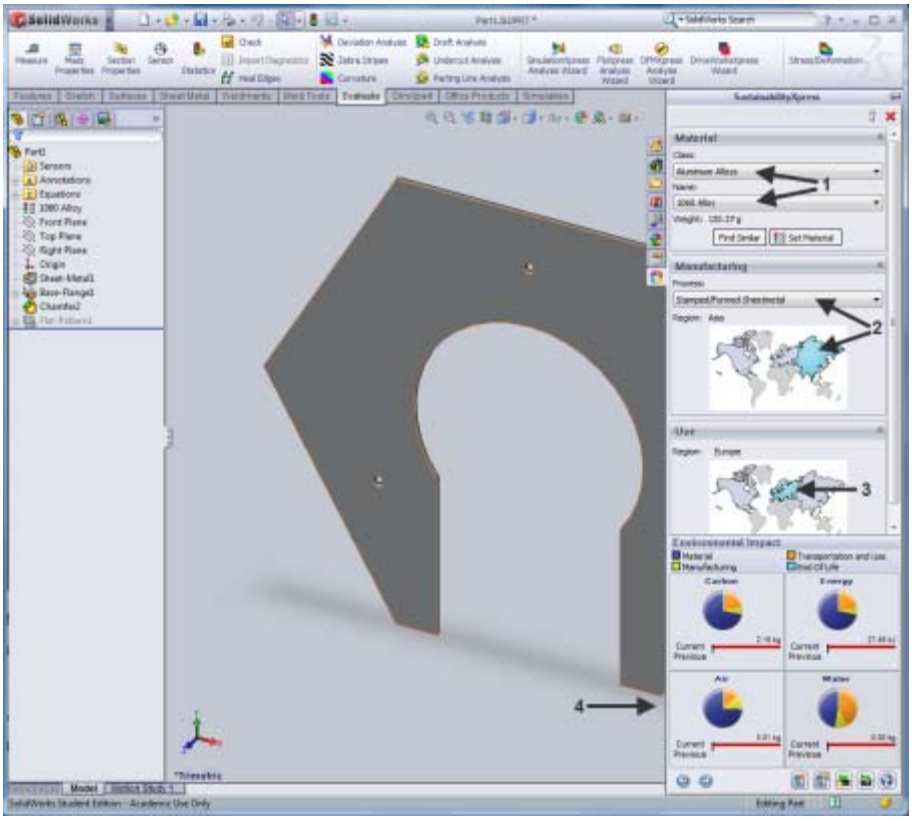
As a developer/designer, you must take several aspects into account.
One of these aspects is the environmental impact of your design.
SolidWorks Sustainability Xpress allows you to understand and visualize the environmental impact of your designs and, if necessary, improve the design.
This includes carbon footprint calculation, ((Footprint) is a measure 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.

27	Before continuing to  , please make sure this SolidWorks function has been installed on your computer. **1.** To open the menu, click the arrow in the Menu Bar:  **2.** Next, click Tools:  **3.** Make sure the function:  has been installed. **4.** If the function: Sustainability Xpress has already been installed, continue to the next step. Otherwise, follow the instructions below.	
		If you are using SolidWorks student Edition version 2009-2010, you must download the application from the following address: http://www.solidworks.com/sw/education/7375_ENU_HTML.htm **Be sure you download the appropriate version, as there are two versions for Windows: 32 bits and 64 bits.**
		If you are using SolidWorks student Edition version 2010-2011, you will find SustainabilityXpress under the tab Evaluate (1) then select SustainabilityXpress (2). 

28	After launching the function: SustainabilityXpress... SolidWorks opens a new window to the right. 	
29	The window includes three important areas: **1.** Material Enter the material properties here. **2.** Manufacturing Specifiy here how and where you want to make the product. Next, enter the destination of the manufactured product. **3.** Environmental Impact Four diagrams are displayed here. They show the environmental impact of production and transportation.	

30 Let us now take a more detailed look at how Sustainability Xpress works.



1. In step 26, the material has already been defined as Aluminum 1060 Alloy. The software copied and pasted this automatically.

2. In the window: Manufacturing you enter this as: Process:

Stamped/Formed Sheetmetal

and for: Region: you select Asia. This defines how you want to manufacture and where the production will take place.

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

4. Under: Environmental Impact 4 diagrams are displayed. They describe the environmental impact of the product.

31

An important part of Sustainability Xpress is the window:

Environmental Impact

As a designer, you can find here various types of information on the environmental impact of your product/design.

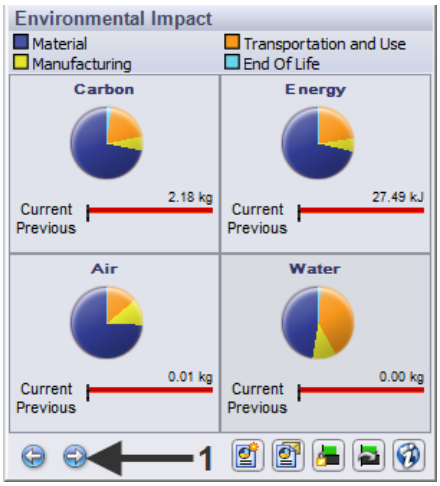
1. Clicking the arrows:  will display four detailed diagrams. They allow you to quickly review the environmental impact during production and transportation of the component.

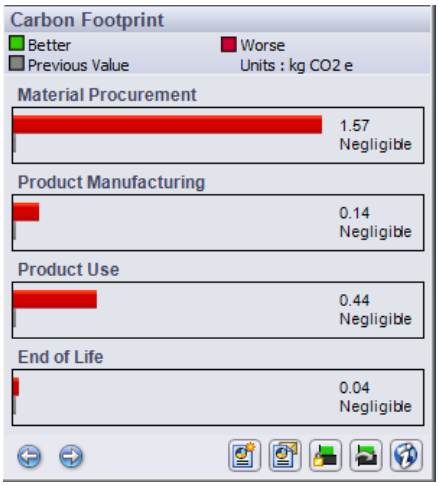
E.g. **Carbon Footprint**
CO2 impact on the environment, e.g. greenhouse gas production.

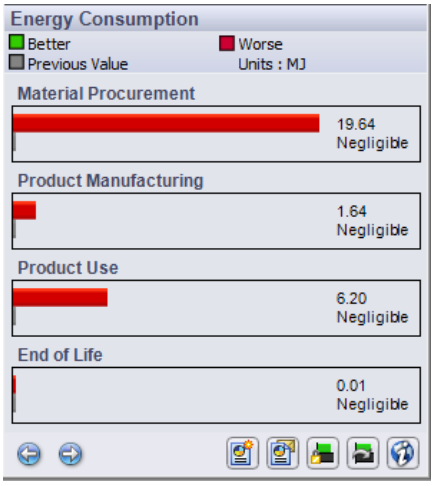
E.g. **Energy Consumption**
The total amount of energy required to manufacture the product.

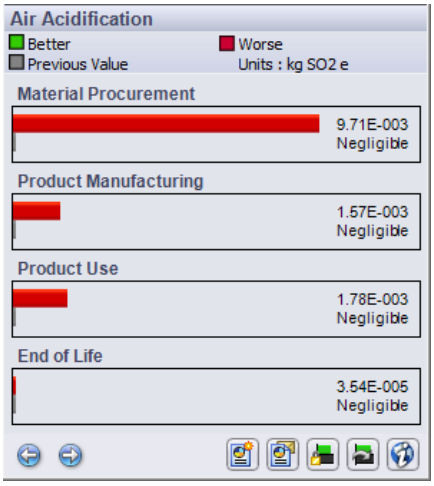
E.g. **Air Acidification**
Impact on the air! In particular, contribution to acid rain.

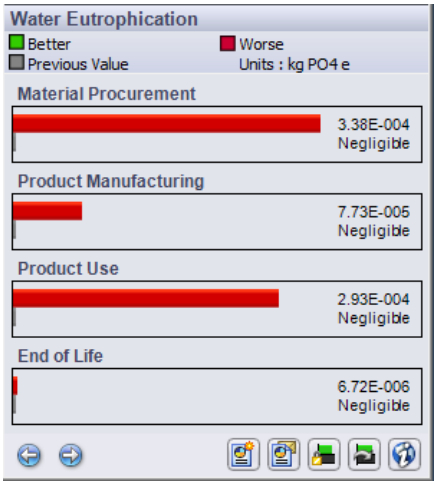
E.g. **Water Eutrophication**
Impact on water! Resulting in algal growth in coastal waters.

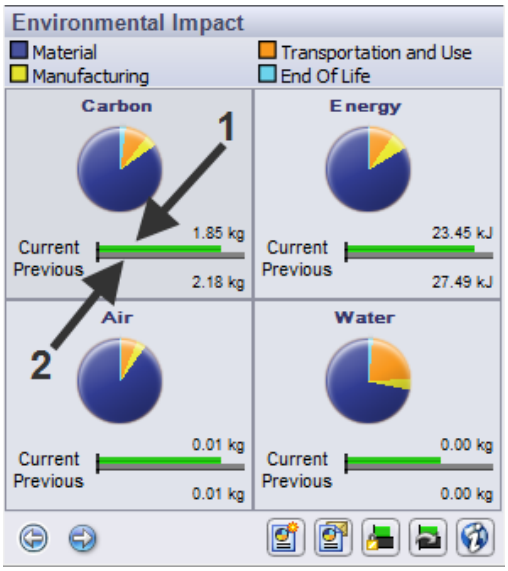










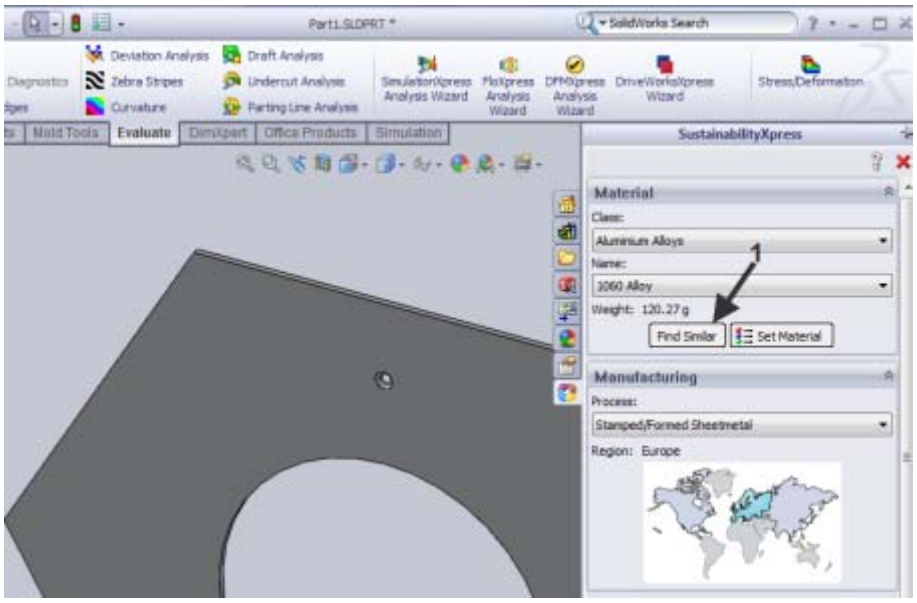
32	Let us now change the production location to see how the environmental impact changes if the base sheet is not produced in Asia but rather somewhere else, for example in Europe. **1.** Change the: Region: into Europe.	
33	Now, watch the diagrams. There is a significant difference between the first and the second calculation. The emission of: Carbon This emission value is now lower than in the first calculation. **1.** Current, (now) is green, meaning: better than the previous location. **2.** Previous, grey represents the first calculation, the previous production location.	

34

But what happens to these values if we choose a different material for the product?

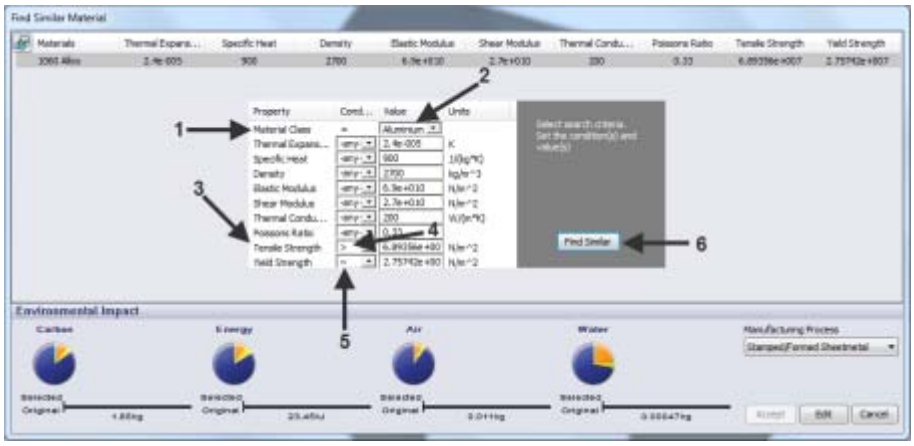
Sustainability Xpress has anticipated that possibility.
Instead of having to search through a very long list of materials, you will see a list of similar materials.

1. Click: Find Similar



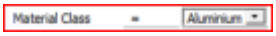
35

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 materials 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.

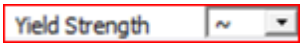
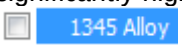
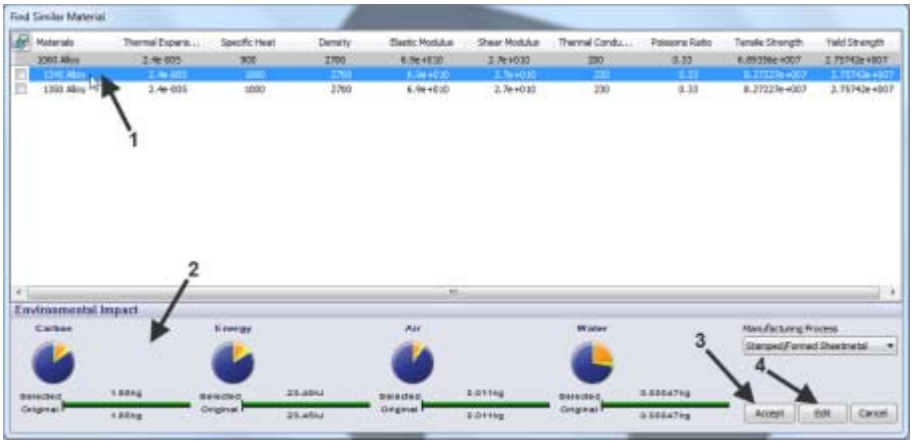


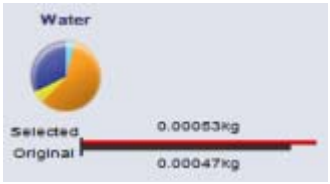
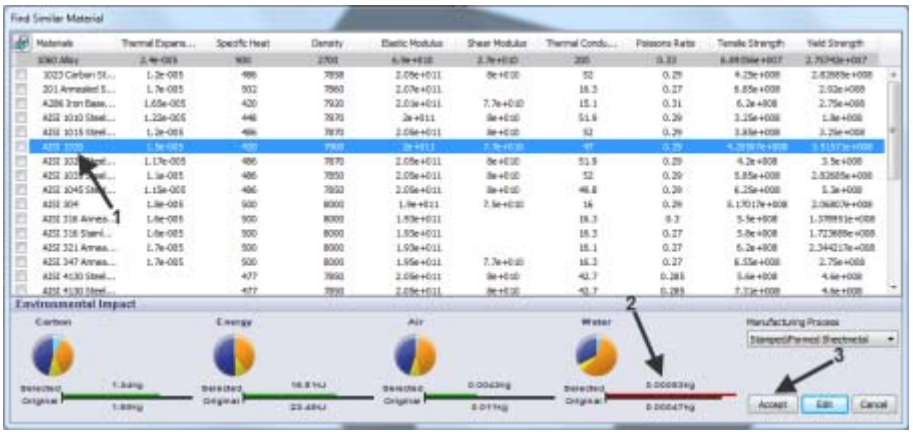
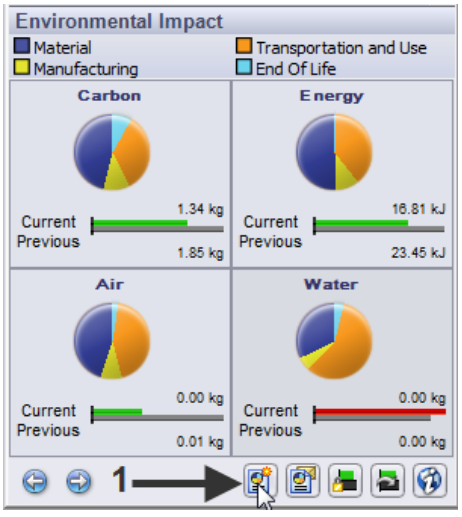
1. Enter Aluminum as the production material.

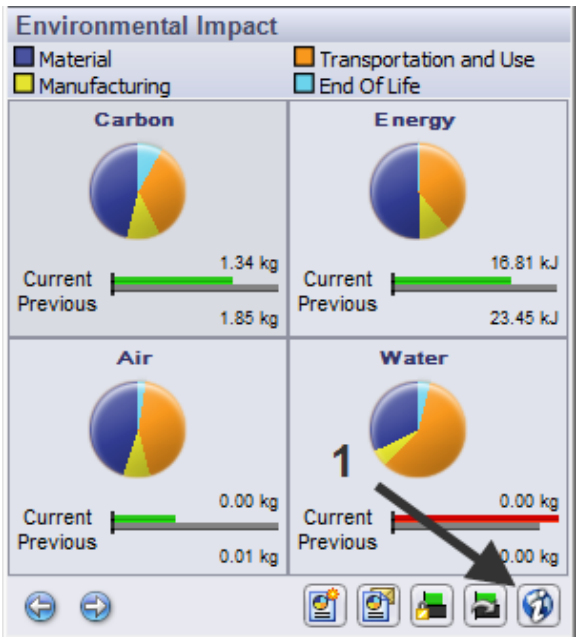
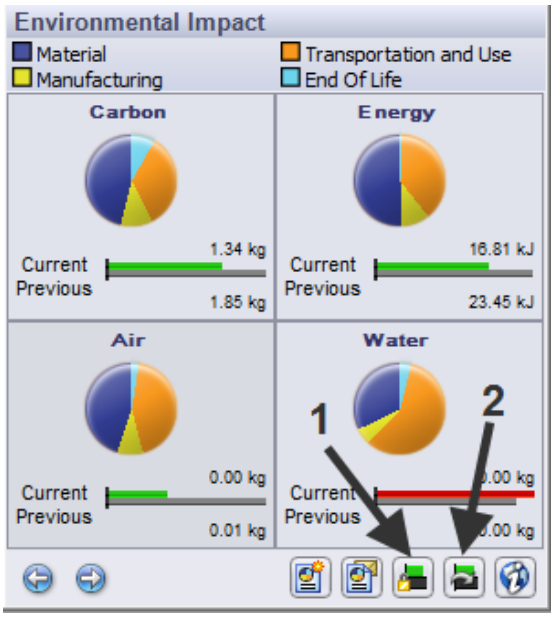
2. Scroll down to choose Aluminum.

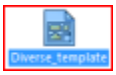
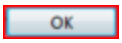
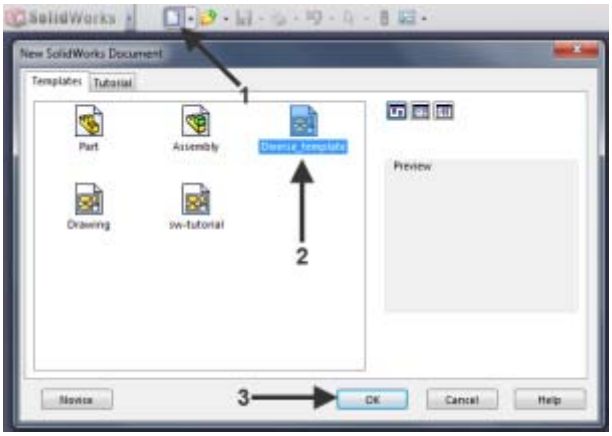
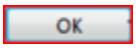
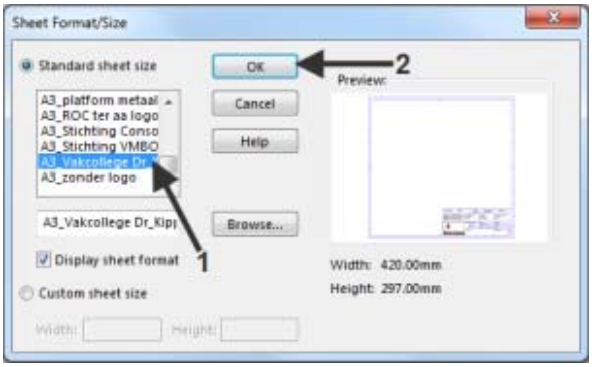


3. The option Tensile Strength is for tensile strength, and Yield Strength is for the yield value.

	4. Let us choose a material with higher tensile strength. Click the scroll down menu and select greater than **>**. **5.** We will leave the yield value of the material unchanged **~**. **6.** Click Find Similar Find Similar	 
36	**1.** Double-click the option 1345 Alloy. This is almost the same material as the one you had chosen (1060 Alloy). There is, however, one important difference: the tensile strength is significantly higher.  **2.** The diagram section immediately displays the new calculation. It is identical to the old one. That is because the material is almost the same. **3.** You can now do the following: Accept, Edit or Cancel.  **4.** Let us choose Edit because we want to know what will happen if we choose steel instead of aluminum.	

37	**1.** Choose Steel as the material. Material Class = Steel **2.** Let us choose a higher tensile strength and a higher yield value. Tensile Strength > Yield Strength > **3.** Click Find Similar. Find Similar	
38	AISI 1020 is a low carbon machine steel offering the good general and structural steel properties. So, we will choose this type of steel. **1.** Double-click: AISI1020. AISI 1020 **2.** The diagram section shows this choice is better for the environment, except for the water.  **3.** Click Accept. Accept	
39	**1.** Click: Generate Report. This button allows you to generate a full report.	

40	The report looks like this. Open the file Sustainability Report.docx to take a closer look. The report allows you to make an informed choice between the original and the alternative material selection.	
41	**1.** If you click: Online Info, online information on CO2 emission during production will be displayed.	
42	**1.** This button allows you to lock the Baseline.  **2.** This button allows you to import the baseline 	

43	Close the function: Sustainability Xpress. **1.** Click the red x **2.** Now try a few other materials yourself to see which is the best solution (e.g. wood).	
44	Now, make a drawing for use in the workshop. **1.** Click New:  **2.** Select: Diverse_template  **3.** Click OK. 	
45	In the menu, choose: **1.**  A3 _Vakcollege Dr_Knippenberg Click OK. 	

46

An empty drawing field is displayed. Do the following to create views.

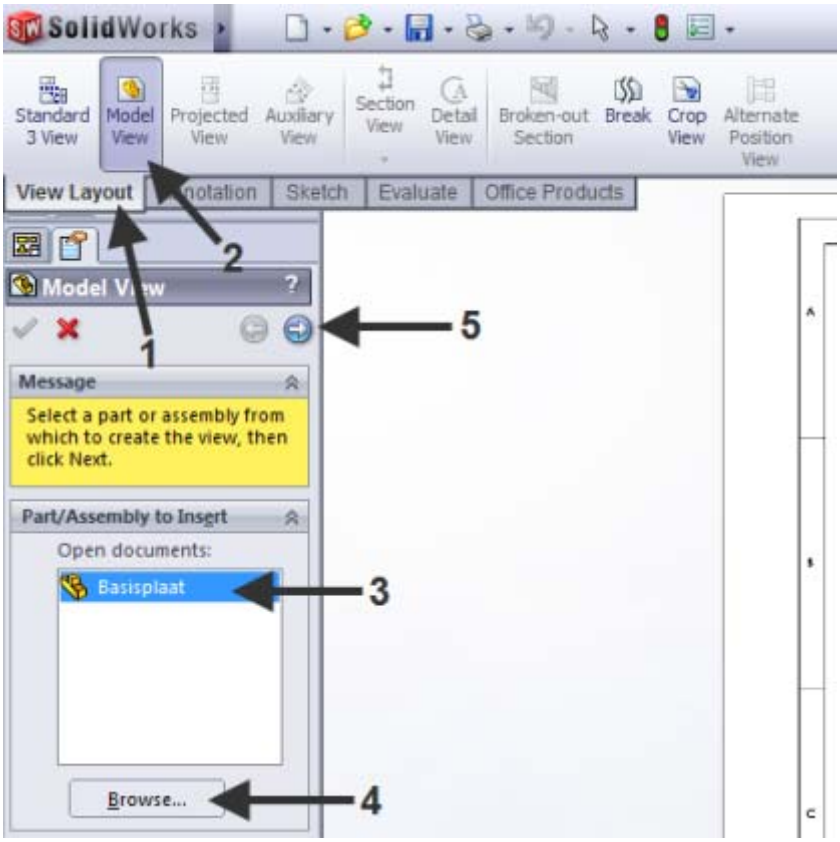
1. Click View Layout

2. Click Model View to open the Property Manager model view.

3. Make sure the appropriate part has been selected.

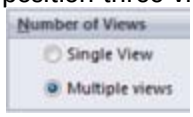
4. If not, use the button Browse... to find the appropriate part.

5. Click the arrow to continue.

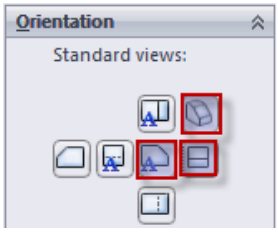


47

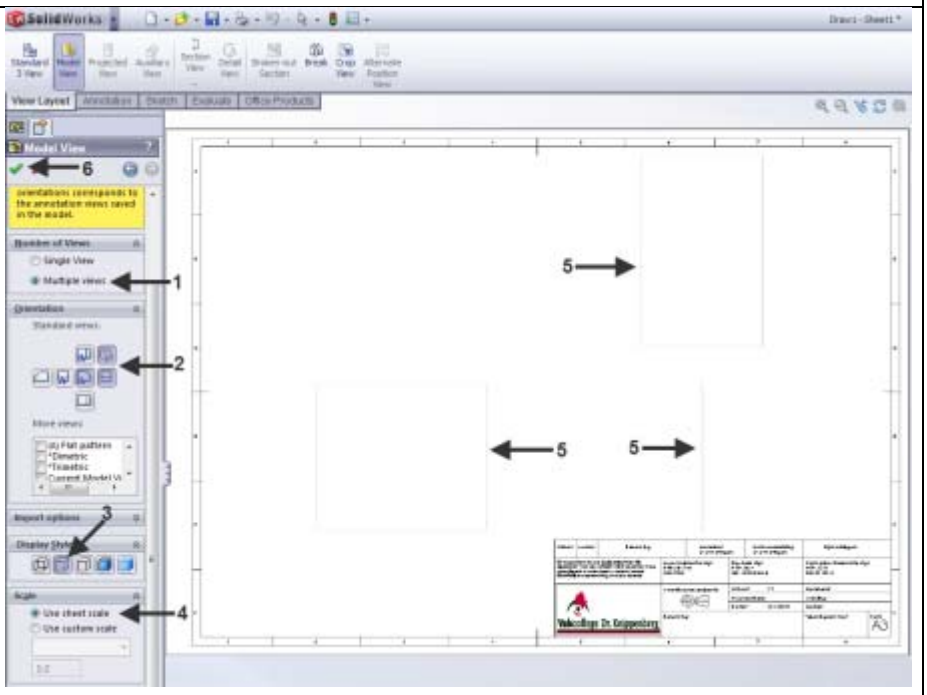
1. In the menu: Number of Views Select: Multiple views. To position three views.



2. In the Orientation menu, choose the front view, the side view and the isometric view. The selected views are displayed in grey.



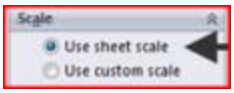
3. For Display Style, choose Hidden Lines Visible.



4. Leave the scale at: Use sheet scale.

5. If all went well, SolidWorks has positioned the three chosen views.

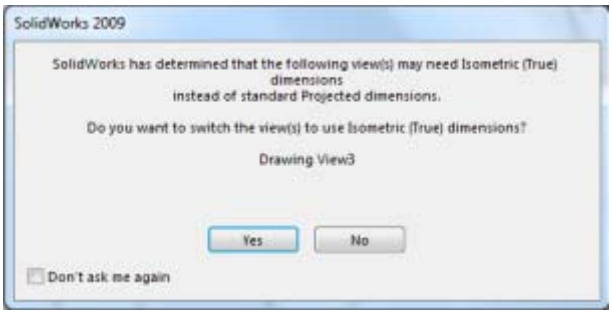
6. Click OK.



Sometimes, a message like in the adjoined figure is displayed.

In that case, click:

Yes

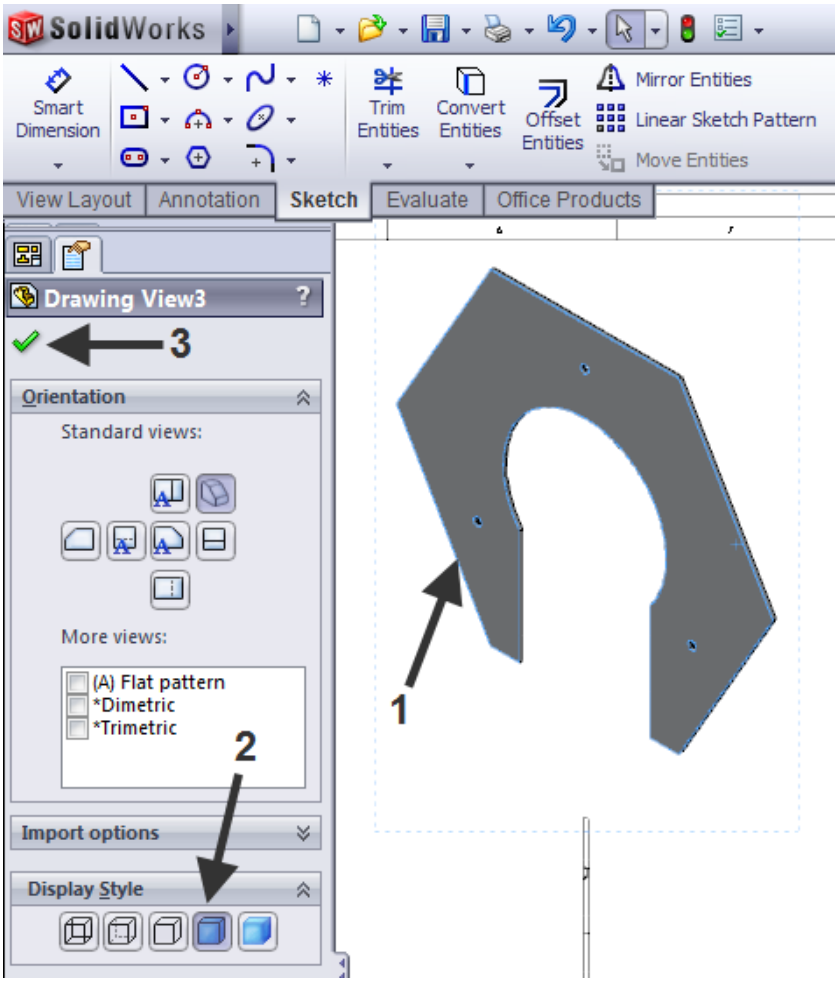


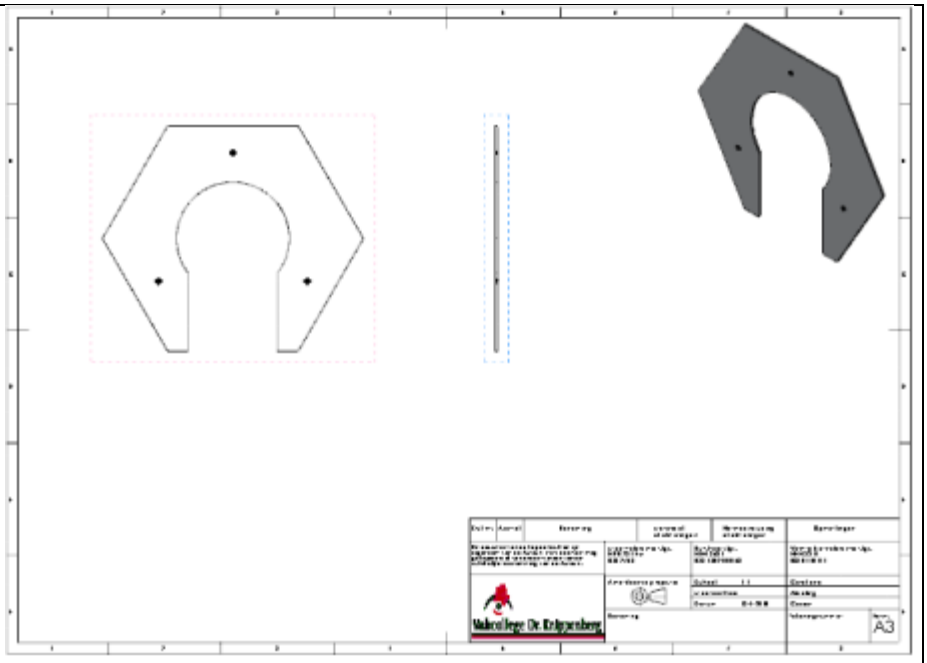
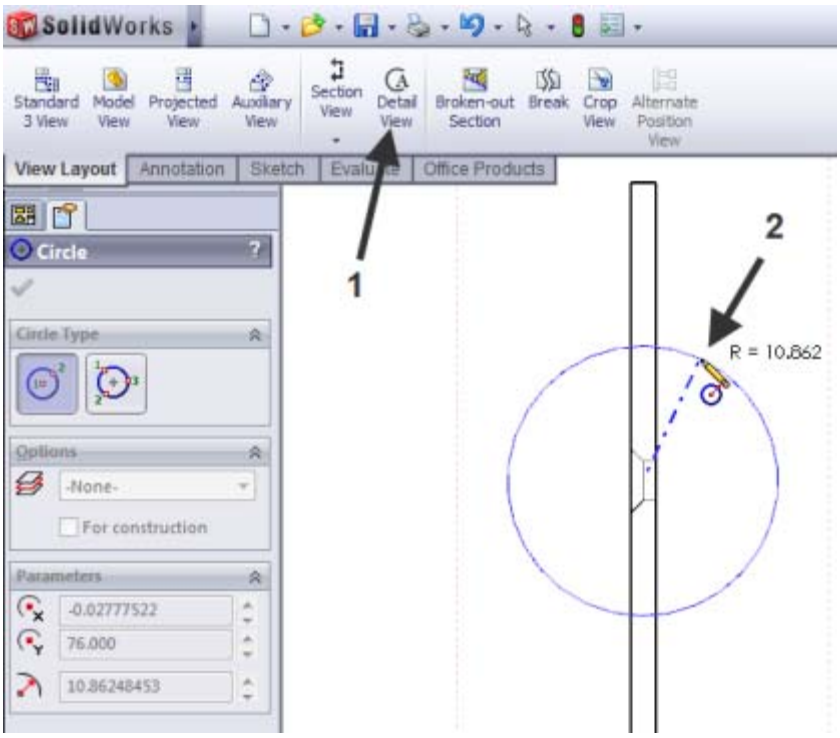
48

1. Now select the isometric view.

2. Click: Shaded With Edges.

3. Click OK.

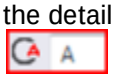


49	Distribute the views on the drawing sheet, as in the adjoined figure.	
50	Add details to the drawing. **1.** Click Detail View. **2.** Draw a circle.	

51

You will have to enter the following information:

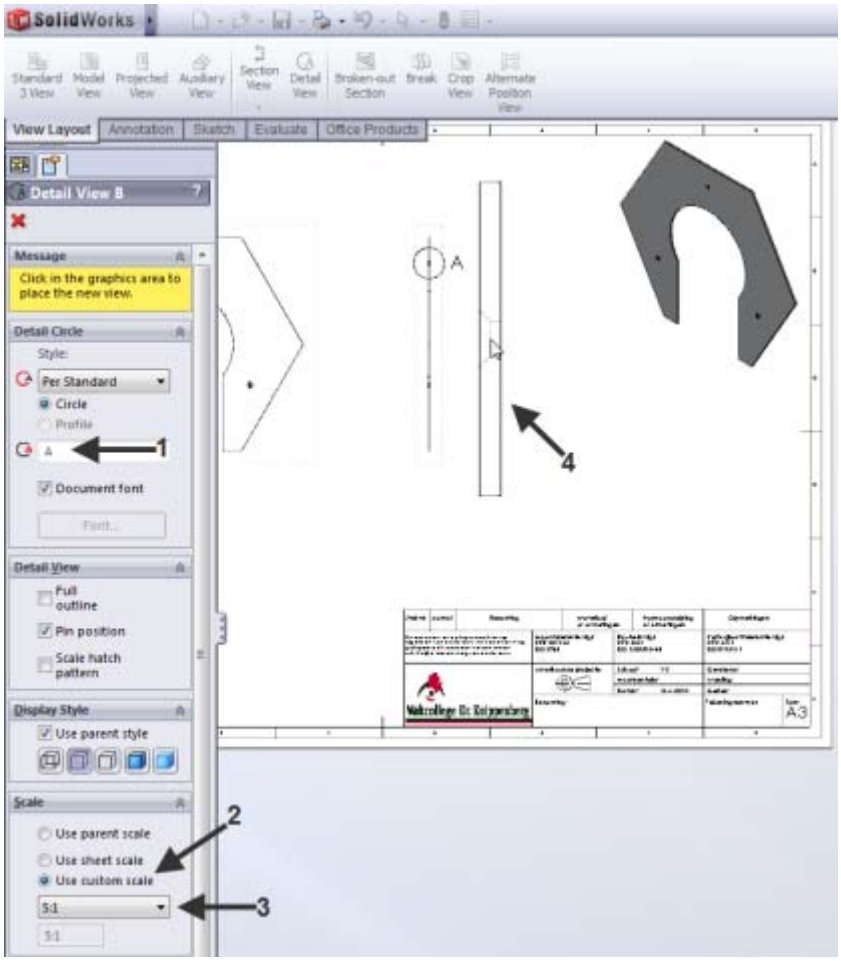
1. Type a capital letter A here to assign an identification mark to the detail.



2. We want to enlarge the detail. Therefore, choose:

3. Enter the ratio

4. Click next to the top view to position the detail.

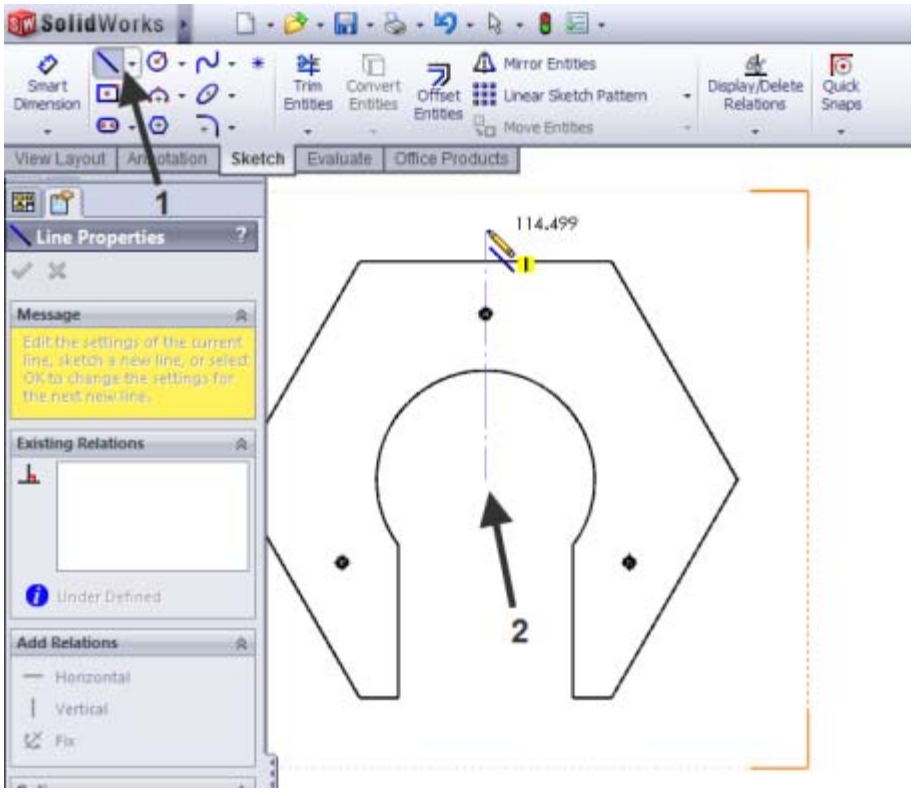


52

Draw a: Centerline from the center of the circle straight up.

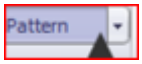
1. Click:  the arrow to open the Centerline function and select it.

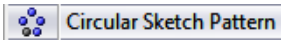
2. Click in the middle of the circle; draw the Centerline, then click anywhere outside the model.



53

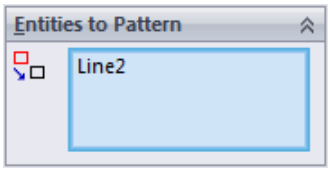
You will add two more lines using the function Circular Pattern.

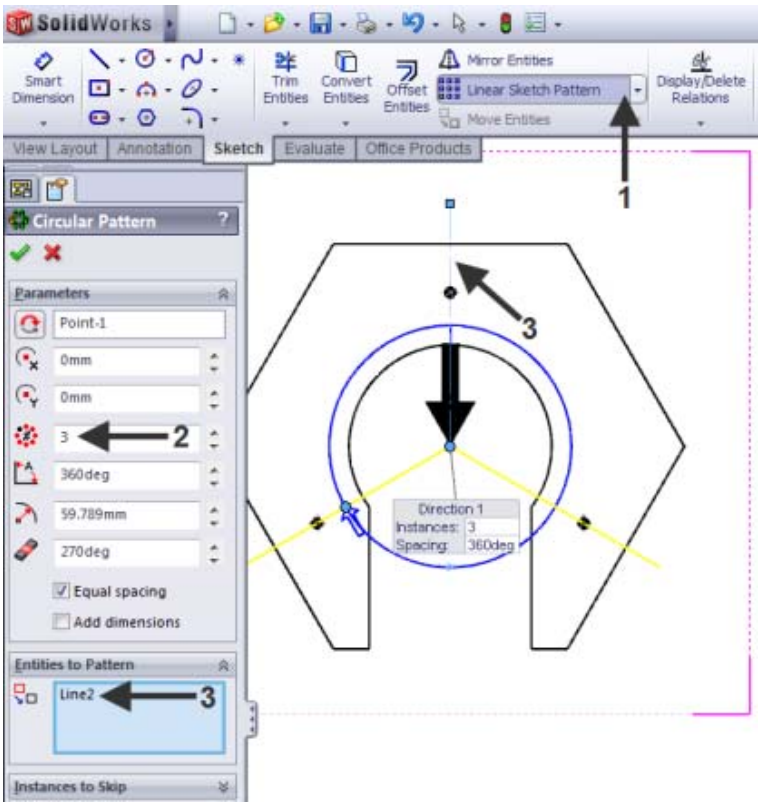
1. Click the scroll down menu 

and select:
Circular Pattern.


2. Enter 3 as quantity.


3. Click anywhere in the window: Entities to Pattern, then select the vertical line you have just drawn.

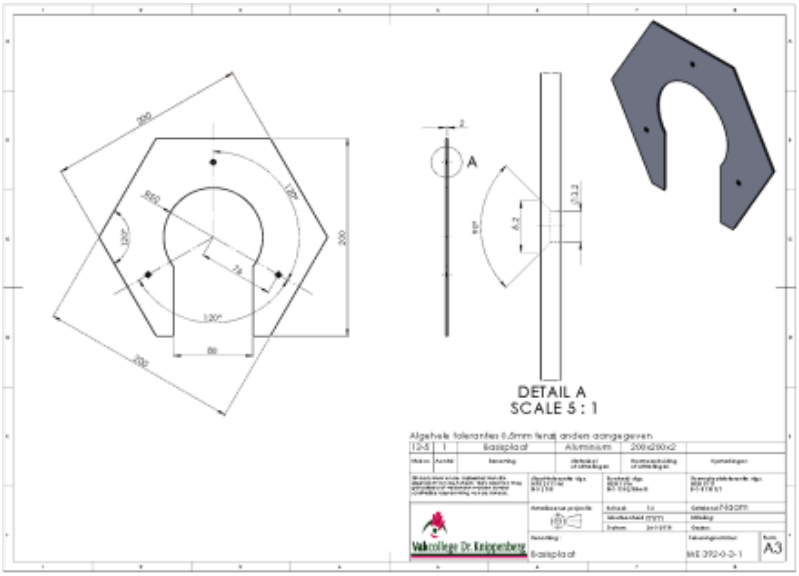




54

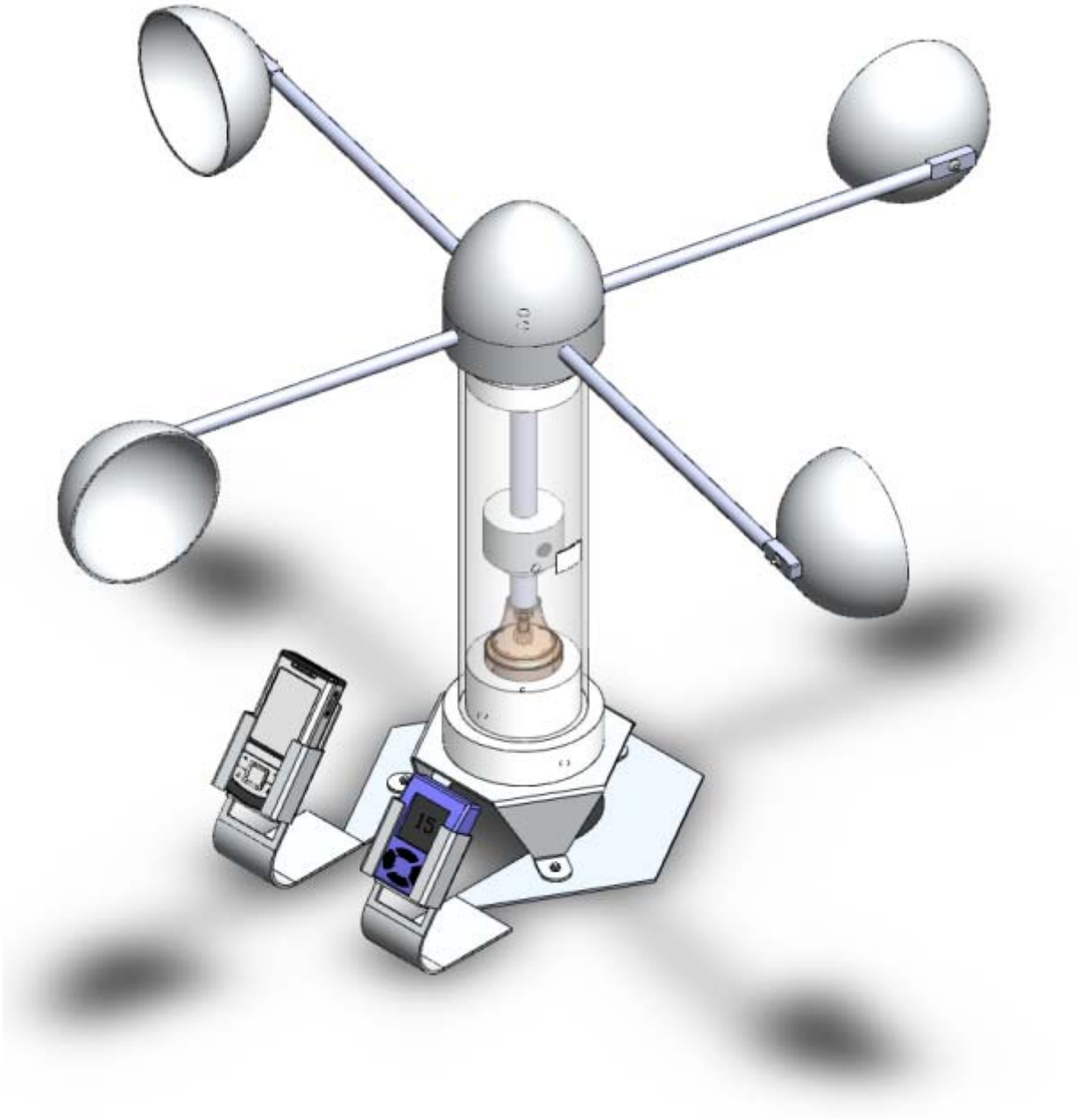
Dimension the drawing as in the adjoined example. Drag the bill of materials (Table) into the drawing and fill out the title block. Copy the data from the example in the adjoined figure.

This completes your working drawing.
Save the file as BaseSheet.slddrw



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

SolidWorks® tutorial 13-12
“Exciting”



Pre-vocational Secondary Education
and Senior Secondary Vocational Education

Stichting Consortium Beroepsonderwijs

SolidWorks

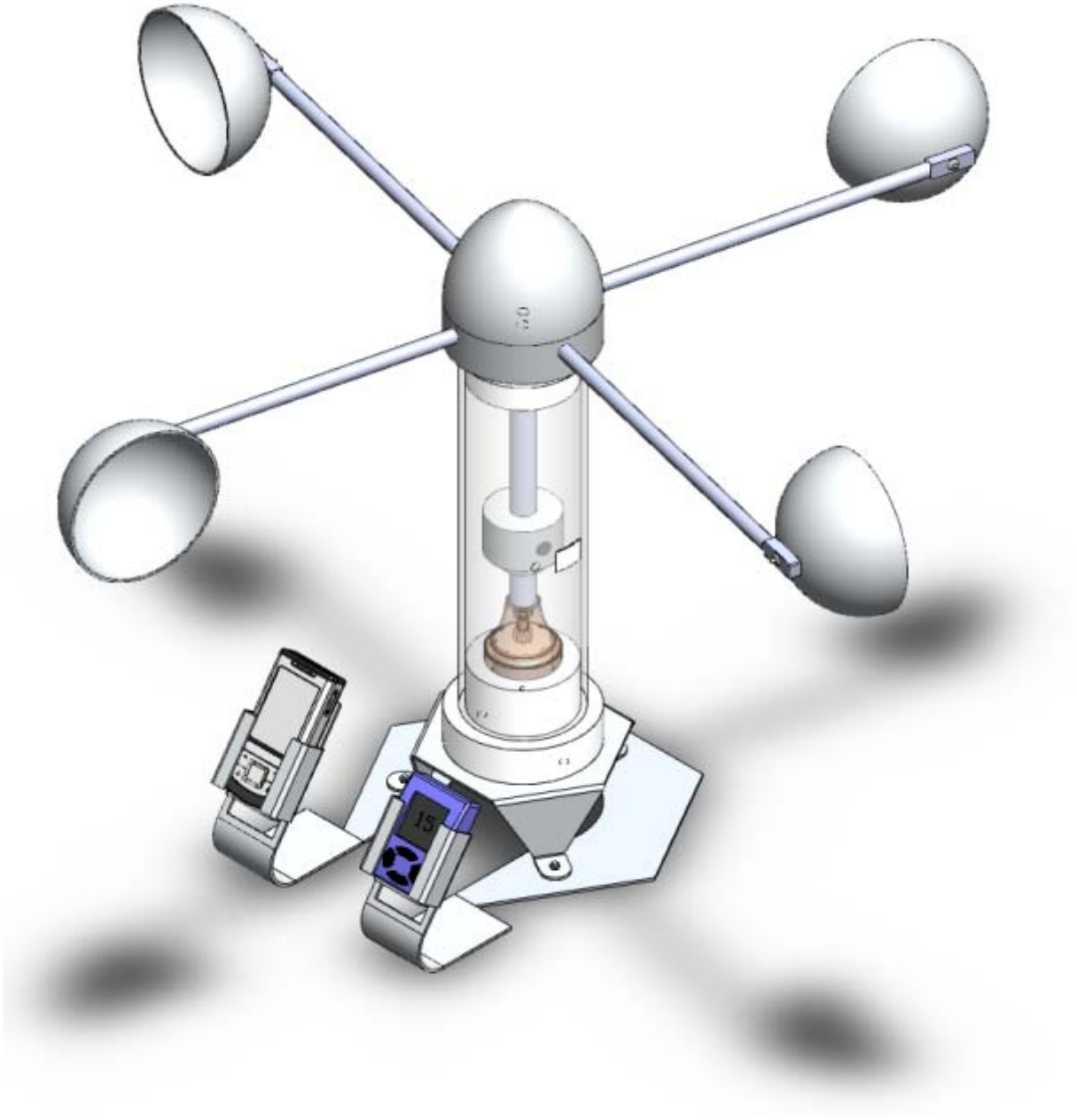
platform metaal & metalektro

For use with SolidWorks® Educational Release 2009-2010

This tutorial was developed for SolidWorks Benelux and may be used by anyone who needs to learn how to use the 3D CAD software SolidWorks. **Any other use of this tutorial or any portion thereof is prohibited.** If you have questions, please contact your retailer.

Intitiative: Jack van den Broek and Nenad Raskovic
Adaptation to the educational level: Jack van den Broek (Technical school Dr. Knippenberg).
Completed by: Nenad Raskovic

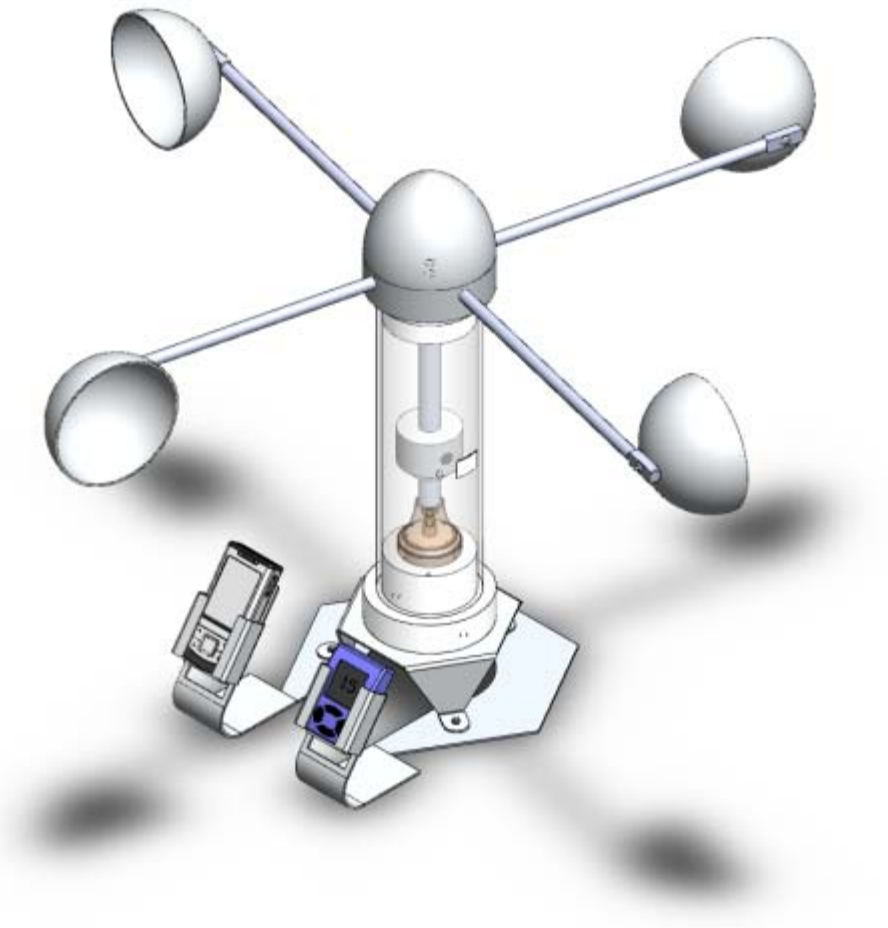
Tutorial 13-12
“Assembling a windmill”



Assembling a windmill

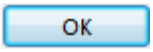
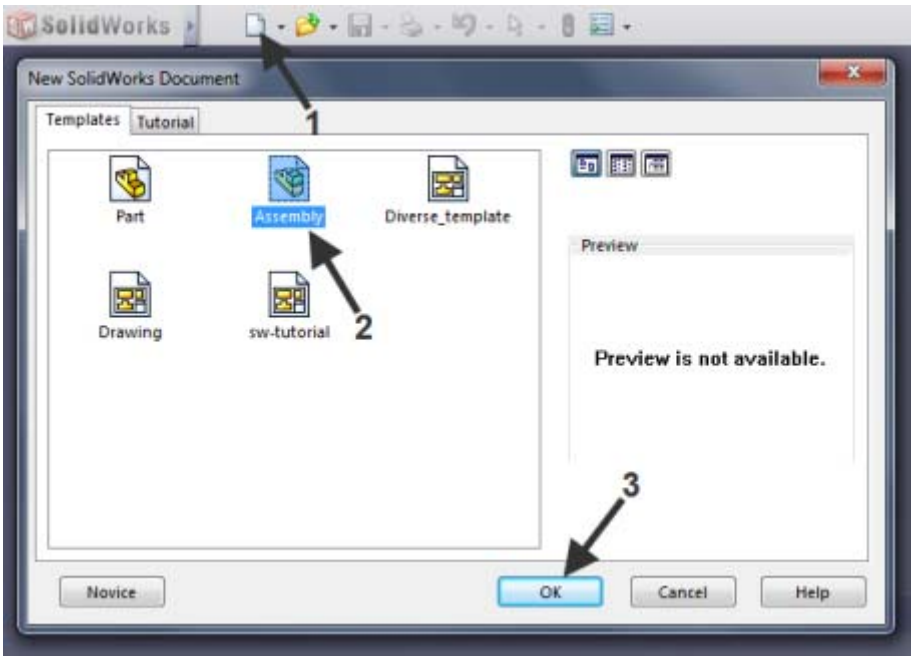
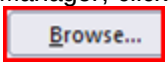
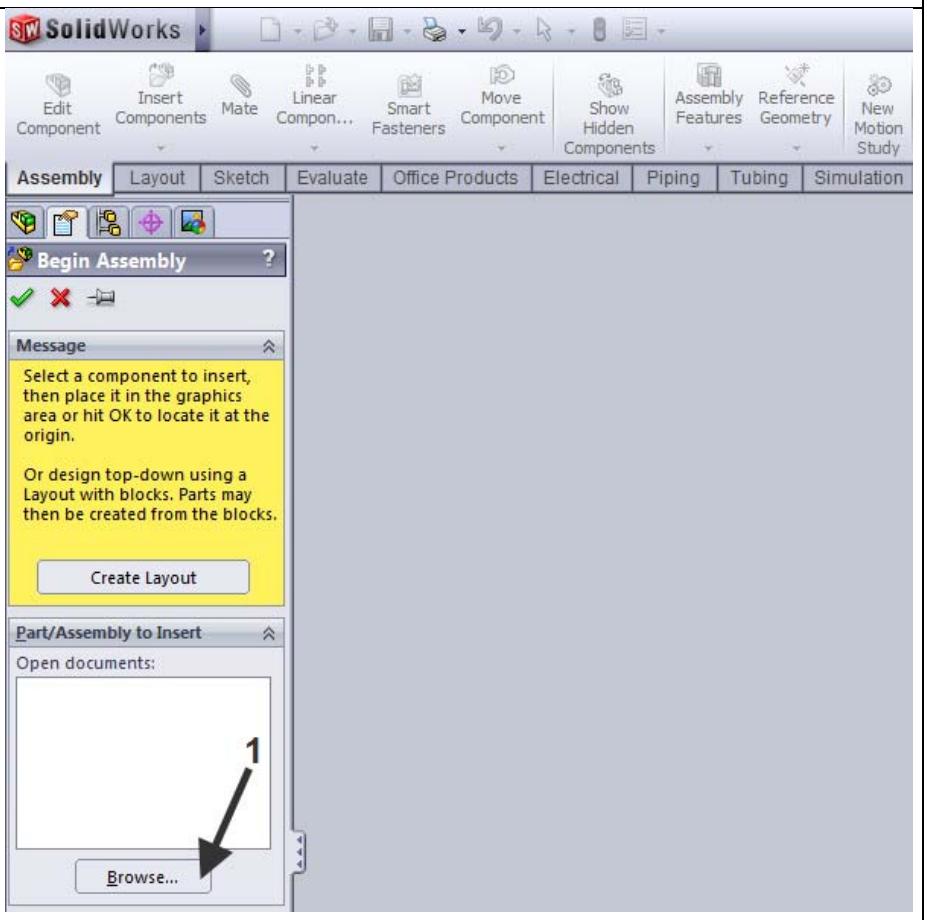
In this exercise, you will get acquainted with assembled products: Assemblies. Assemblies consist of all pieces you will have made in previous tutorials, together with a couple of pieces you will have to purchase. In this Tutorial, you will learn how to connect one piece to another,

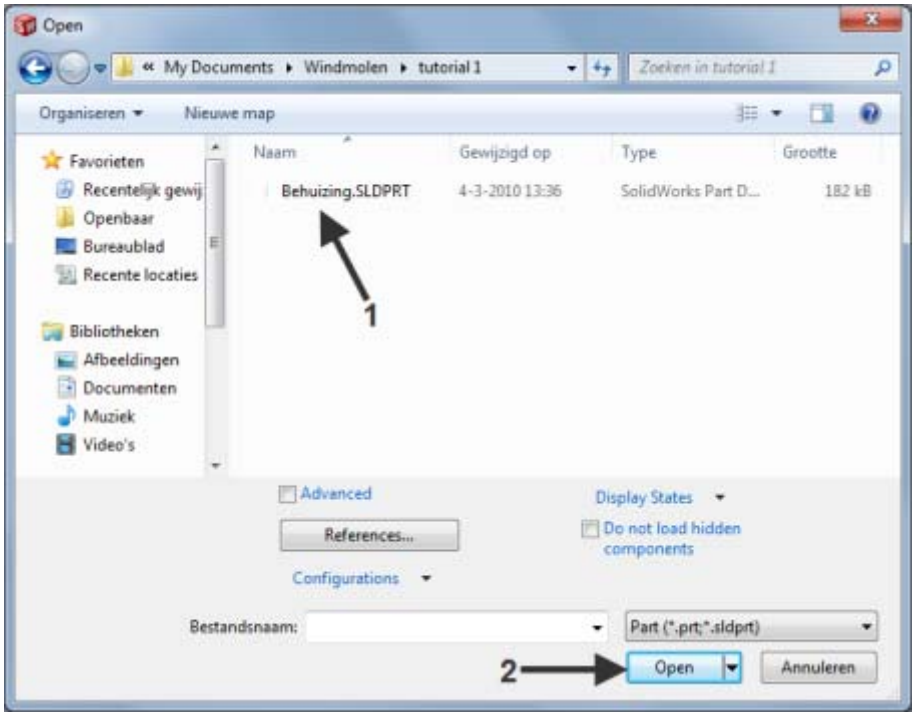
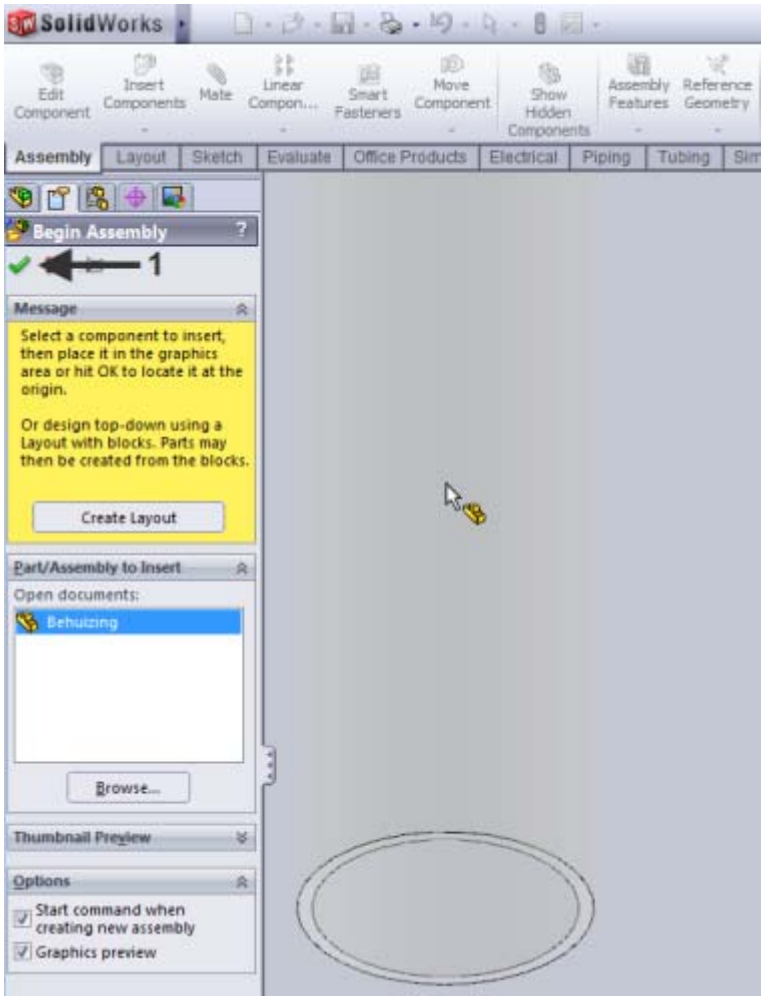
Work plan

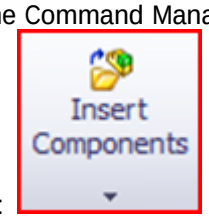
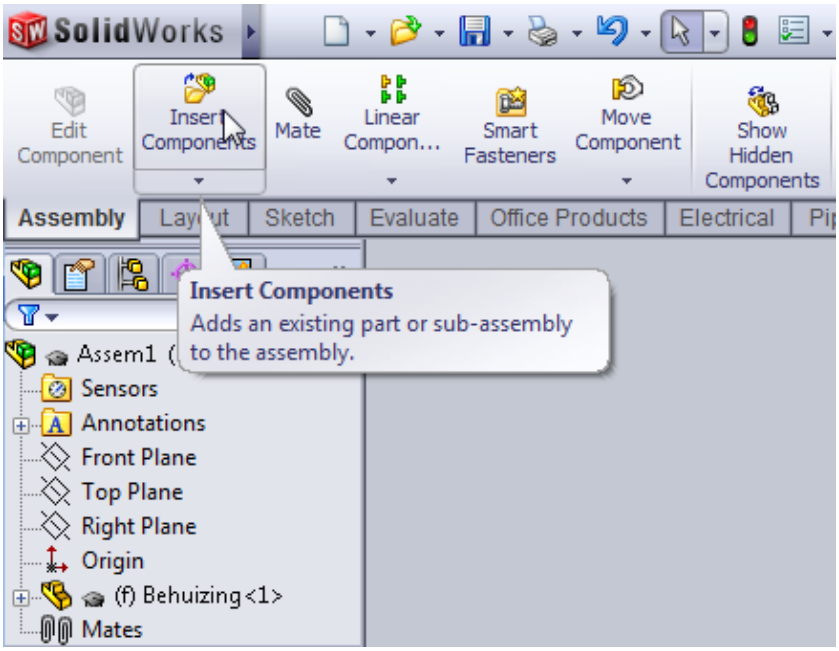
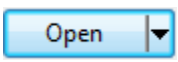
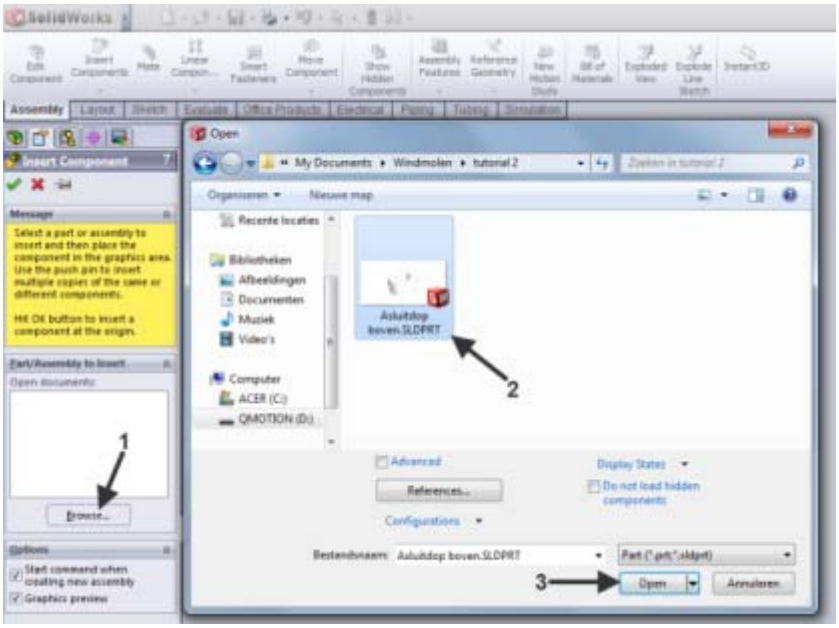


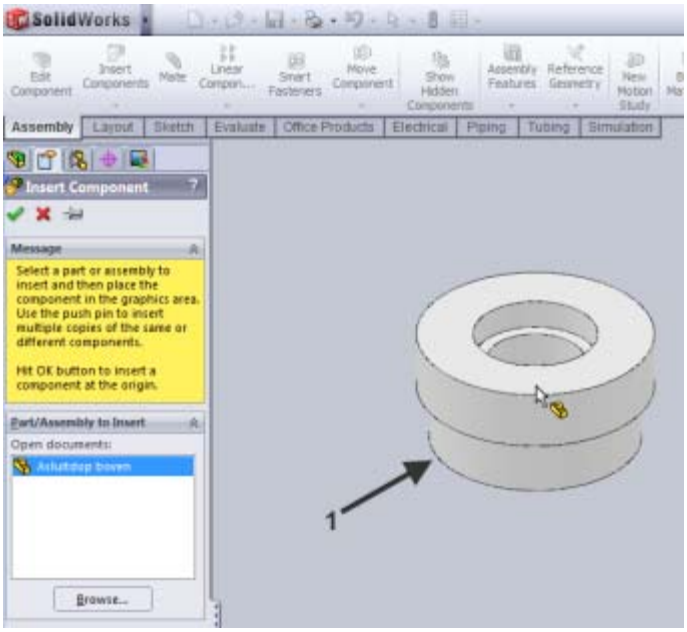
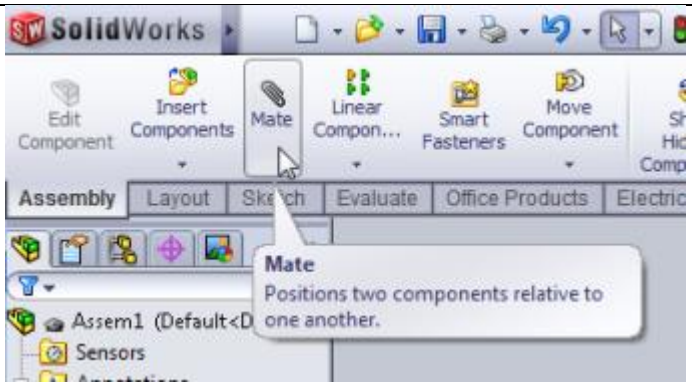
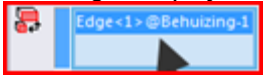
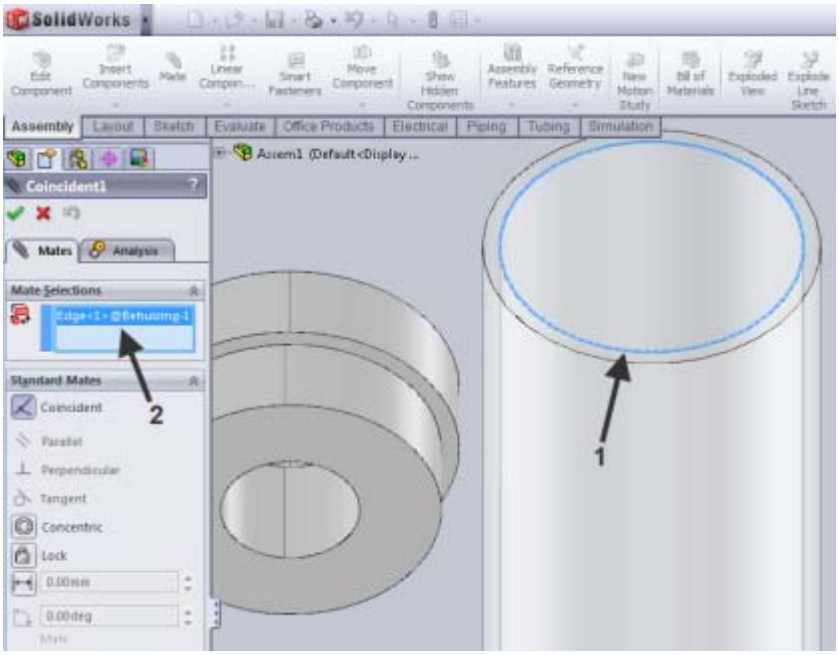
You will assemble a windmill. You will use pieces you have made yourself and pieces that have to be purchased.

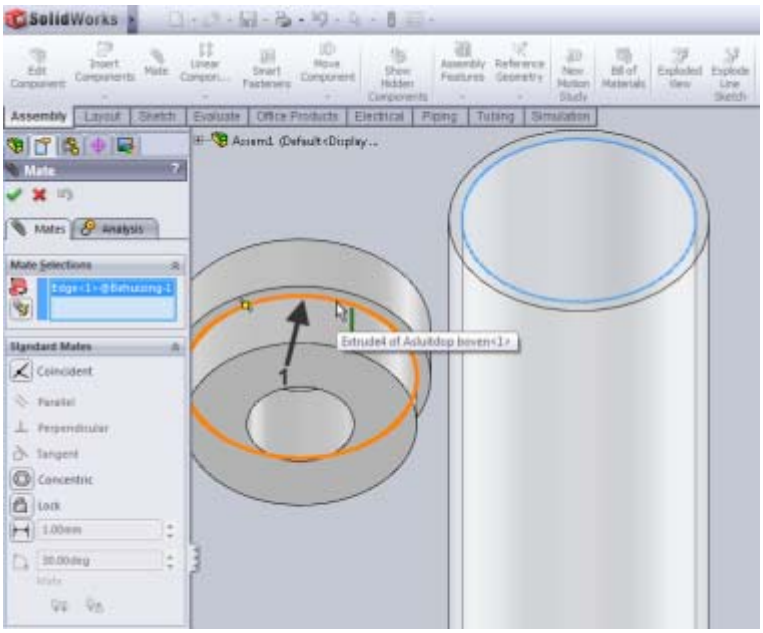
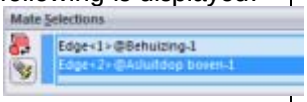
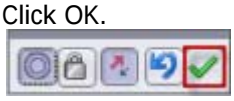
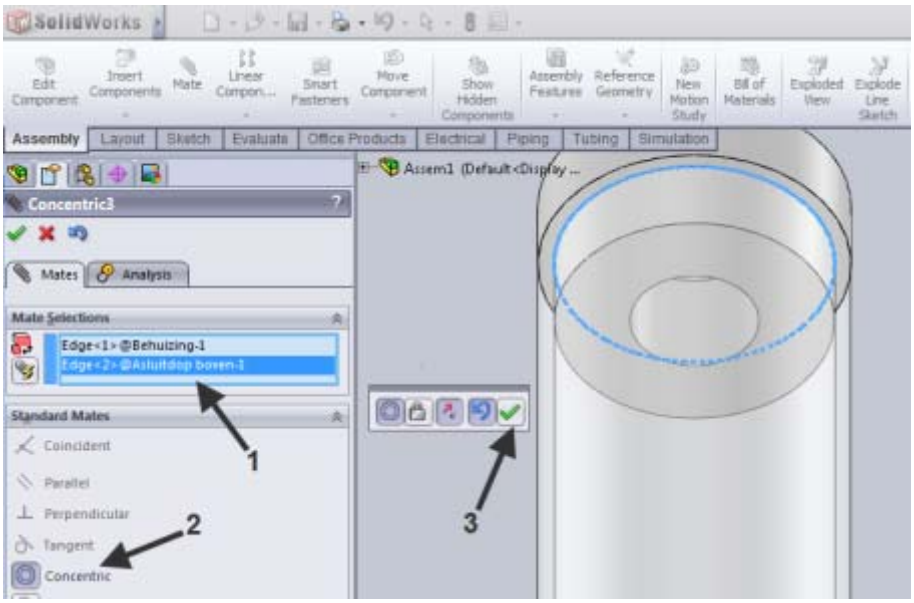
- First, you will learn how to bring pieces into the Assembly environment.
- Next, you will learn how to assemble the pieces (**mate**).
- You will learn how to use the **Toolbox**.

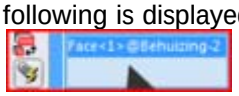
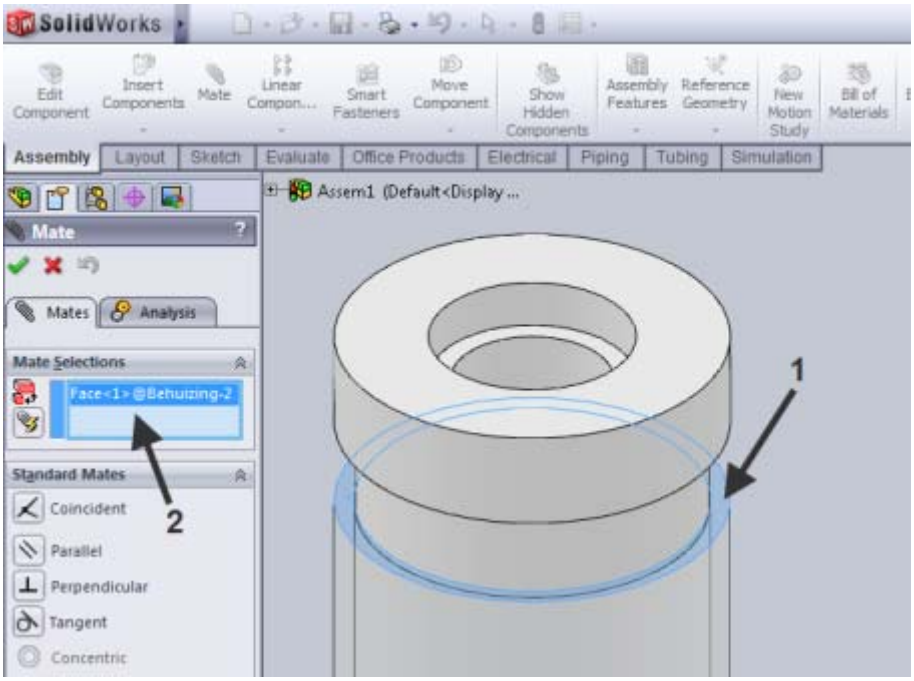
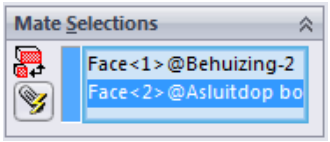
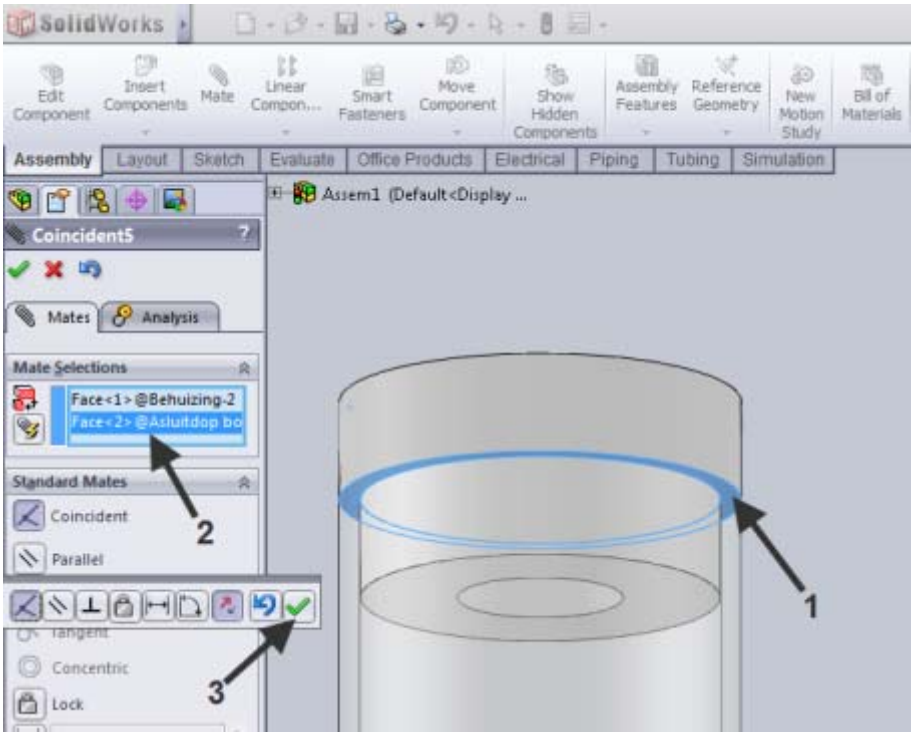
1	Launch SolidWorks.  **1.** In the menu bar, click New.  **2.** In the displayed menu, select: 'Assembly'.  **3.** Click OK.  	
2	A new menu appears. **1.** In the Property Manager, click:  Using this action button, search for the appropriate piece. We are looking for  The first piece we made. **Attention**: This step may work differently as described here. If so, read the instructions below.	

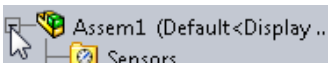
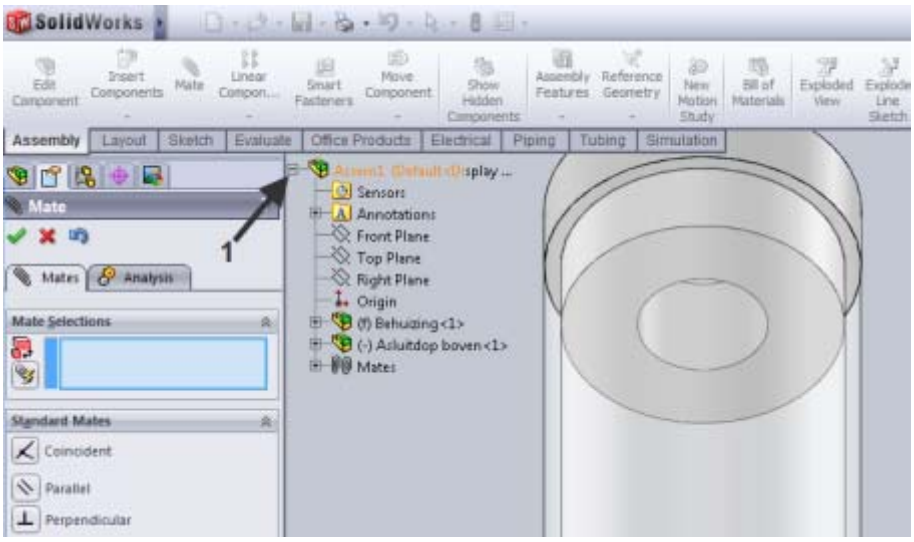
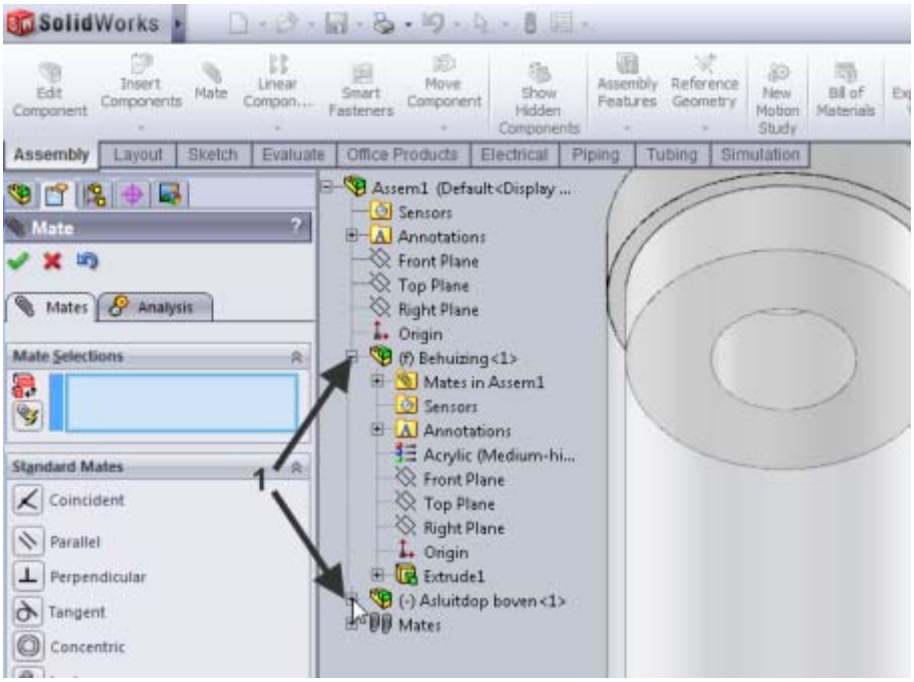
3	Go to the folder in which you saved your models. **1.** Select the model **Housing.SLDPRT**  **2.** Next, click Open 	
4	The piece is now hanging to the cursor.  **1.** Click OK:  to place the piece at the origin.  Placing the piece at the origin is very important for a correct assembly of the entire product.	

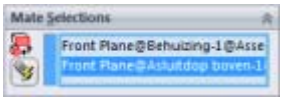
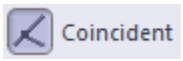
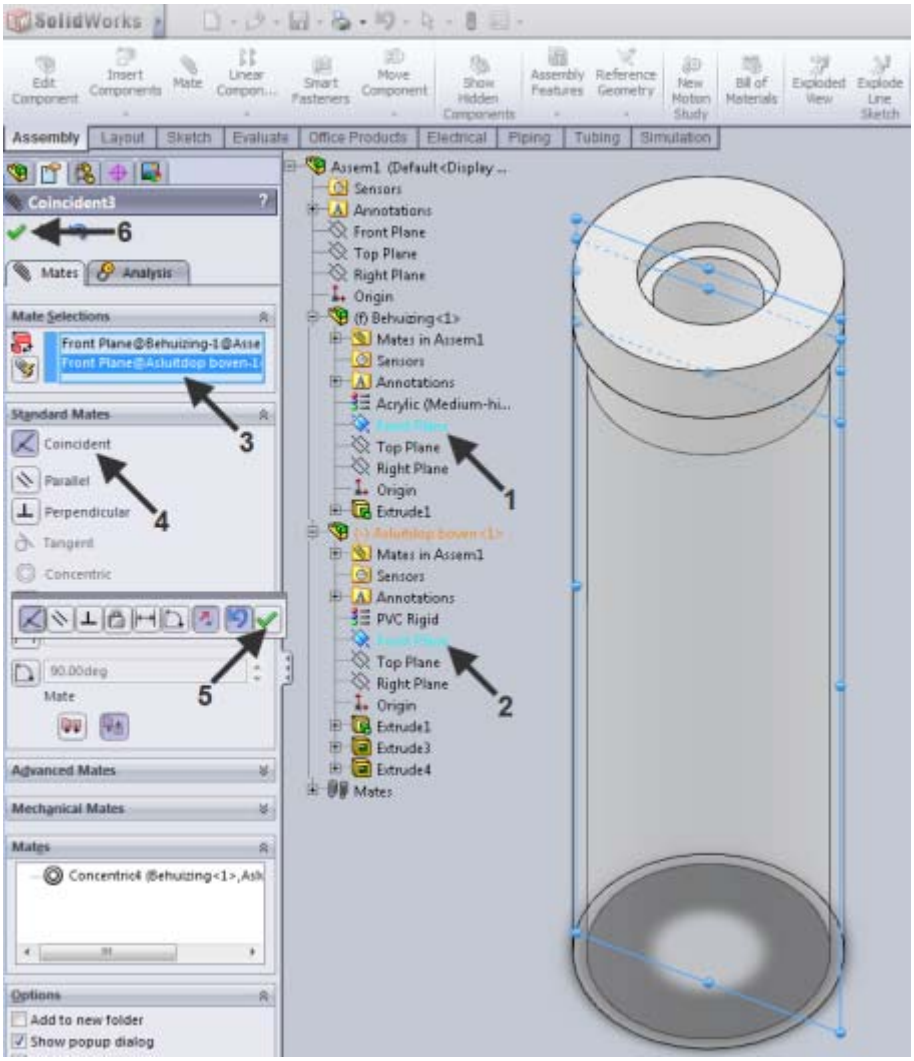
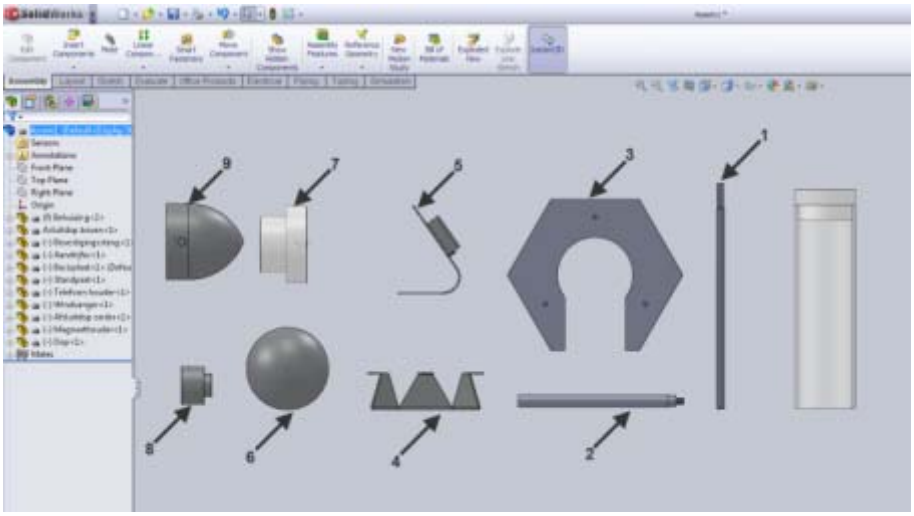
		In the next step, there will be a few things which might not work in the same way as described above. • If the left column is very different from the example above, the command 'Insert Components' did not start automatically. In that case, click 'Insert Components' in Command Manager. • If the pieces are on the list, you apparently had left it open. In that case too, click 'Browse' and start searching for the required document. (Housing), Next, you can simply put it in the assembly as we did above.
5	In the Command Manager, click:  to add the next piece to the Assembly.	
6	We will begin searching for the new piece in the file. **1.** Click:  **2.** Next, select the piece **Cap_Internal.SLDPRT**  **3.** Click Open 	

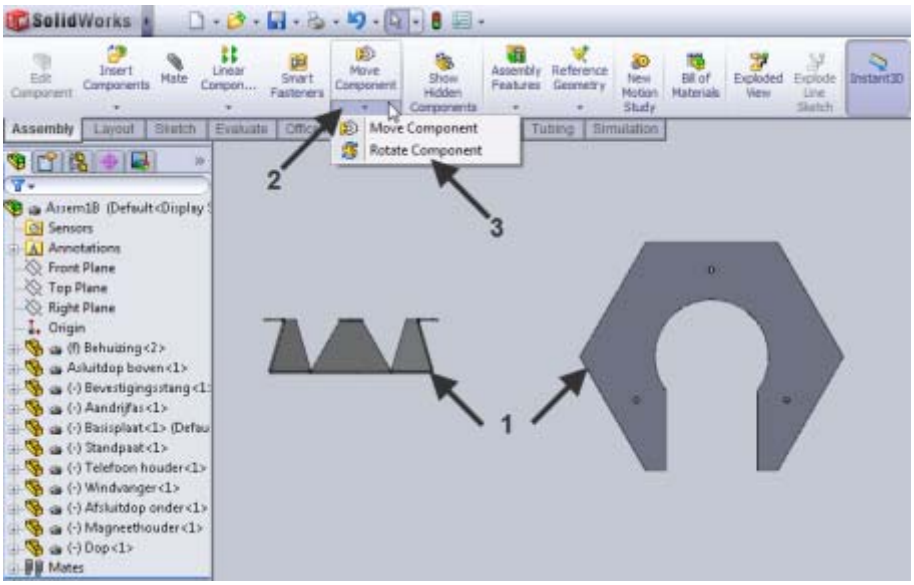
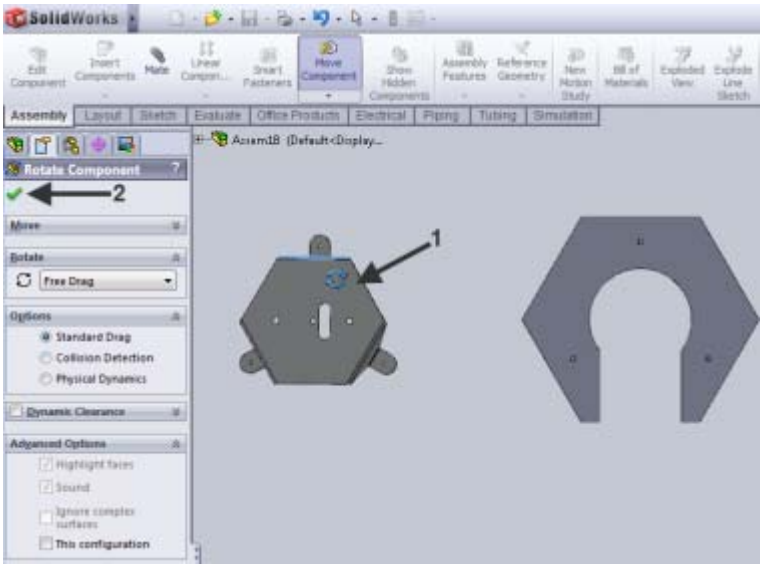
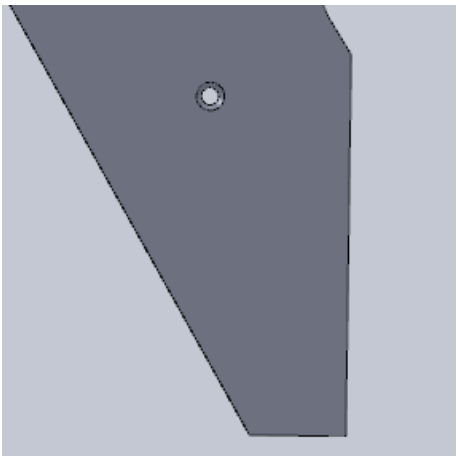
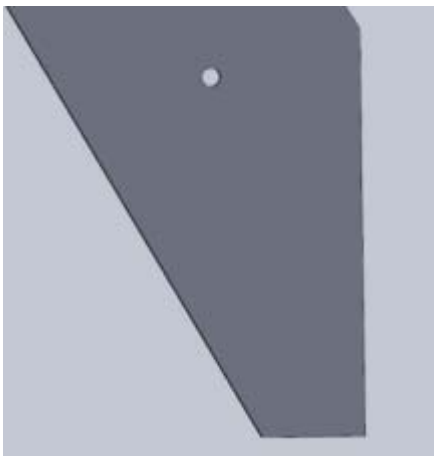
7	**1.** Click anywhere in the drawing area to add the piece.  The added piece is now positioned at an arbitrary location in the assembly.	
8	Now, we will connect both pieces: **together**. **1.** In the: Command Manager, click: 	
9	You must now select two elements between which a 'Mate' will be made. This needs to be done very carefully. Zoom in on the top section of the housing. **1.** Select the inner **edge** of the hole (an 'Edge' and not a 'Face' (Face = flat)). **2.** In the blue area of the Property Manager, the following is displayed: 	

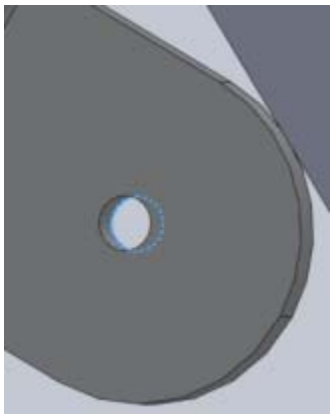
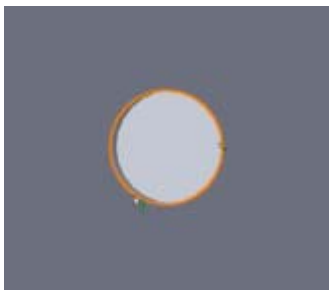
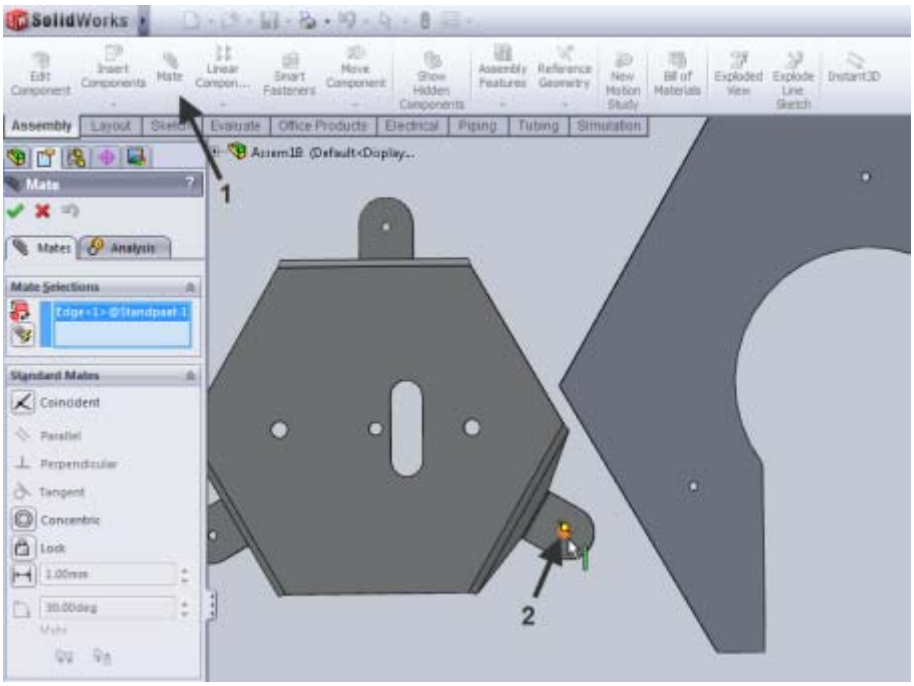
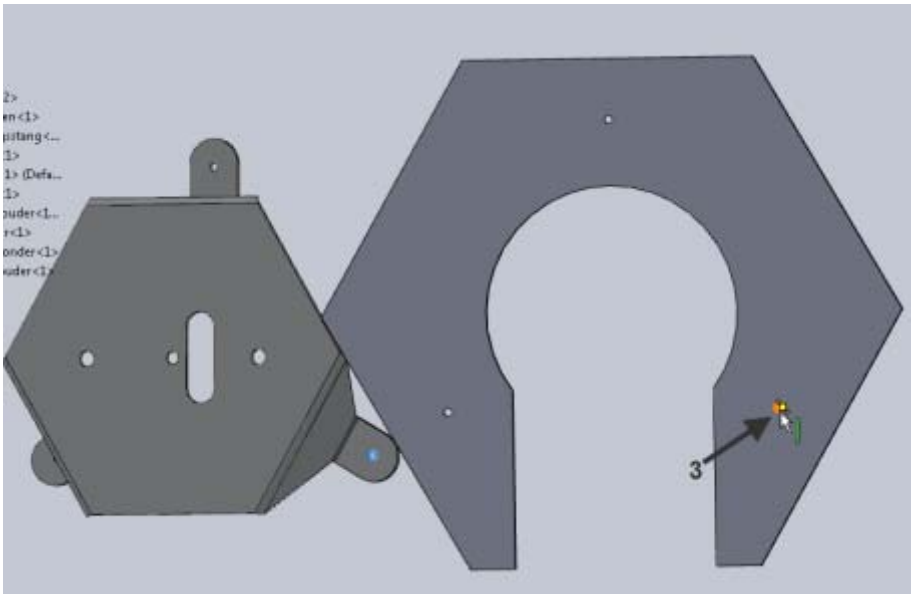
10	Rotate the model so the bottom of the **Upper cap** becomes visible. To achieve this, press the scroll wheel of the mouse or use the arrows on the keyboard. **1.** Select the edge of the **Upper cap**, as illustrated. Again, make sure you did not select a face.	
11	Both parts now move toward each other. **1.** In the blue area of the Property Manager, the following is displayed:  **2.** SolidWorks has already selected the appropriate 'Mate'.  **3.** Click OK. 	
		You may accidentally choose the wrong face or edge. In that case, do the following: With the right mouse button, click in the blue area Mate Selections  Then click Delete to remove the selected part (displayed in dark blue in the window). Click Clear Selection to remove everything.

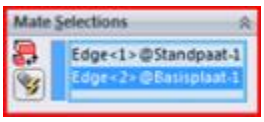
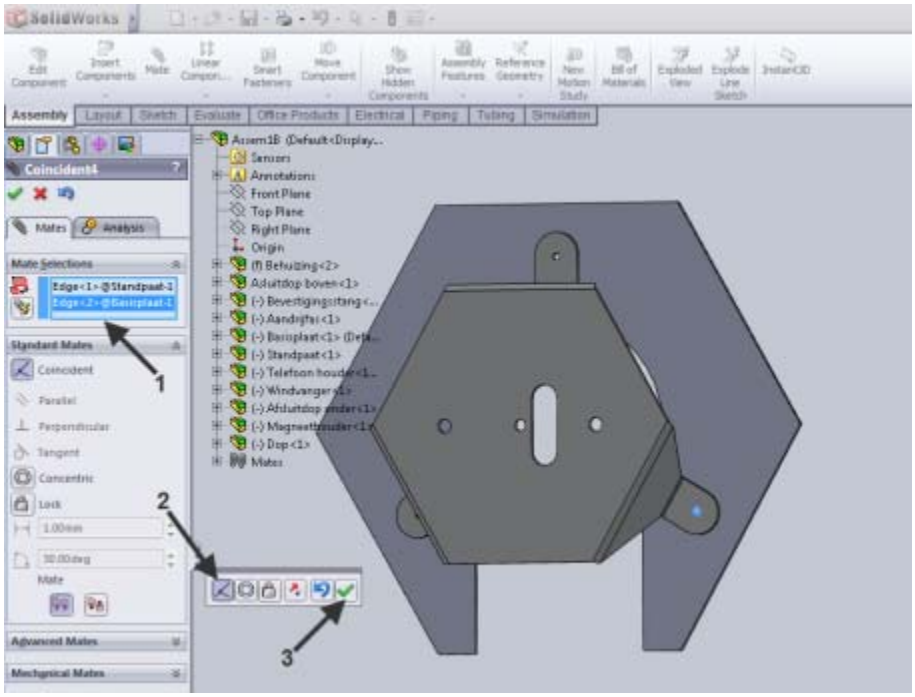
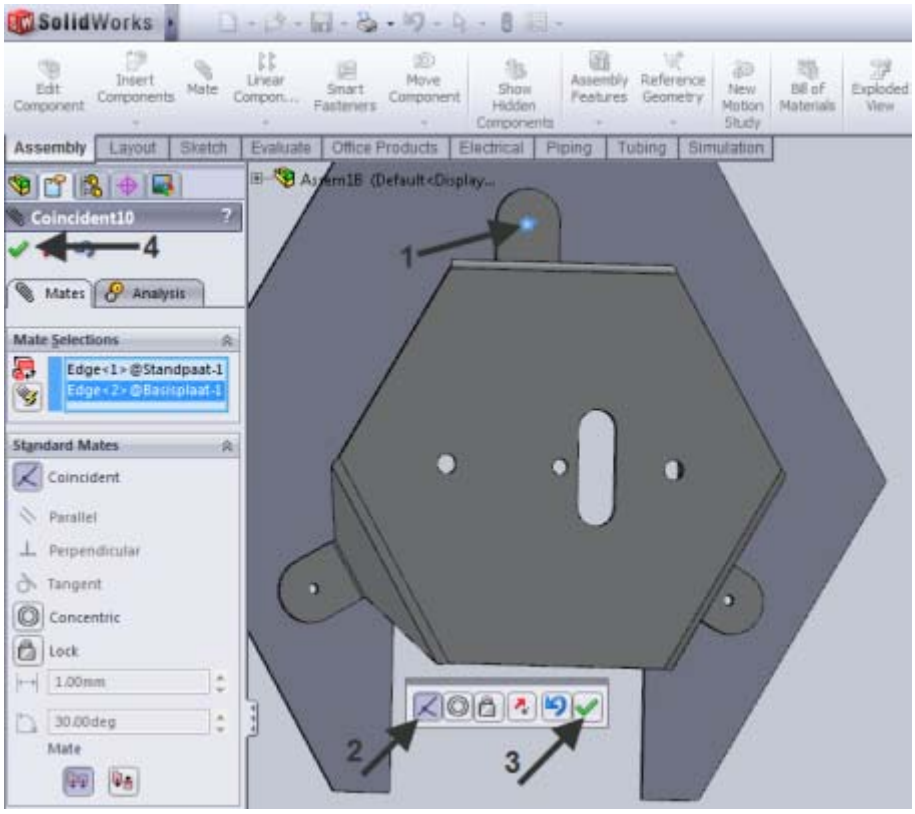
12	If necessary, move the upper cap as illustrated. **1.** Select the face of the hole (a 'Face' and not an 'Edge'). **2.** In the blue area of the Property Manager, the following is displayed: 	
13	Both parts now move to each other. **1.** In the blue area of the Property Manager, the following is displayed: Face<1>@Housing-1 Face<2>@Cap Internal-1  **2.** For 'Mate' , SolidWorks has selected  this time **3.** Click OK to confirm the 'Mate'. 	

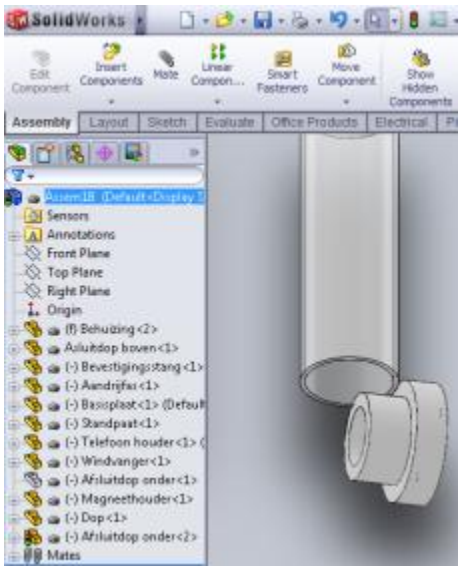
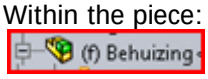
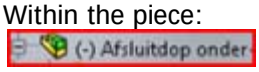
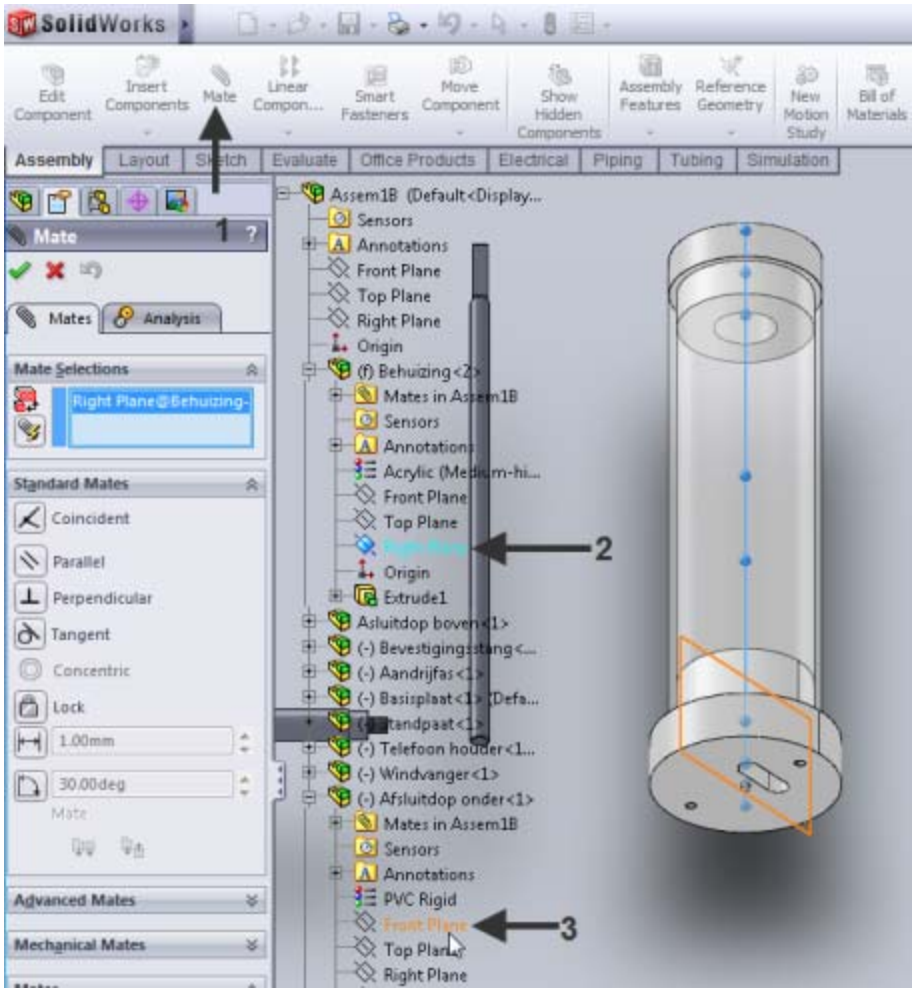
14	The selection area in the: Property Manager is emptied, so you can immediately enter the next mate. To fasten the cap, we use the Front Plane standard face. However, it cannot be selected in the model, but only in the Feature Manager. Since the Property Manager and not the Feature Manager is displayed, you have to use the Feature Manager displayed in the diagram area. **1.** Click the plus sign in front of the file name. 	
15	**1.** Click the plus signs for both pieces. **Attention!** After having clicked the first +, the list scrolls.	

16	**1.** Within the **housing element**, select the  **2.** Within **upper cap**, also select the  **3.** The selected pieces are displayed in the blue area of the Property Manager.  **4.** As 'Mate', SolidWorks has selected  **5.** Click OK to confirm the 'Mate'.  **6.** Click OK once more to close the Property Manager.	
17	Now, add the other required pieces. You can do that by repeating step 5 through 7. Be sure to do that in the following order. 1. Wing Arm 2. Shaft 3. Base Sheet 4. Housing Base 5. Phone Holder 6. Windblade 7. Cap Internal 8. Magnet Holder 9. Top End	

18	We will now continue 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 	
19	**1.** Rotate the **housing base** as in the adjoined figure. **2.** Click OK.	
20	Zoom in at the **base sheet** to see on which side the recessed holes are located. If visible, rotate the piece in such a manner that the recessed holes will be on the bottom side.	 Not like this!!  But like that!!

21	**1.** Click:  **2.** Zoom in on the **<u>housing base</u>** and select the **lower** edge of the hole.  **3.** For the **<u>base sheet</u>**, select the **upper** edge of the hole. 	 

22	Once you have clicked the upper edge of the hole, parts start moving toward each other. **1.** The selected pieces are displayed in the blue area of the Property Manager.  **2.** As 'Mate', SolidWorks has selected  . **3.** Click OK: 	
23	**1.** First, select the lower edge of the hole of the **housing base**. Then select the upper edge of the hole in de **base sheet**.  **2.** Coincident  is the appropriate mate, so we'll leave it like that. **3.** Click OK:  **4.** Click OK once more to exit the function.	

24	Drag the piece: **Cap Internal** toward the housing. See the example opposite.	
25	Mate! **the housing** and **the cap internal** as in steps 8 through 13. **1.** Click:  **2.** Within the piece:  now select the  **3.** Within the piece:  then select the   If necessary, click the plus sign to open the list of both pieces.	

26

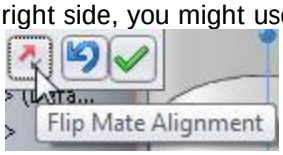
The Front Plane of the **bottom end** now turns towards the Right Plane of the **housing**.

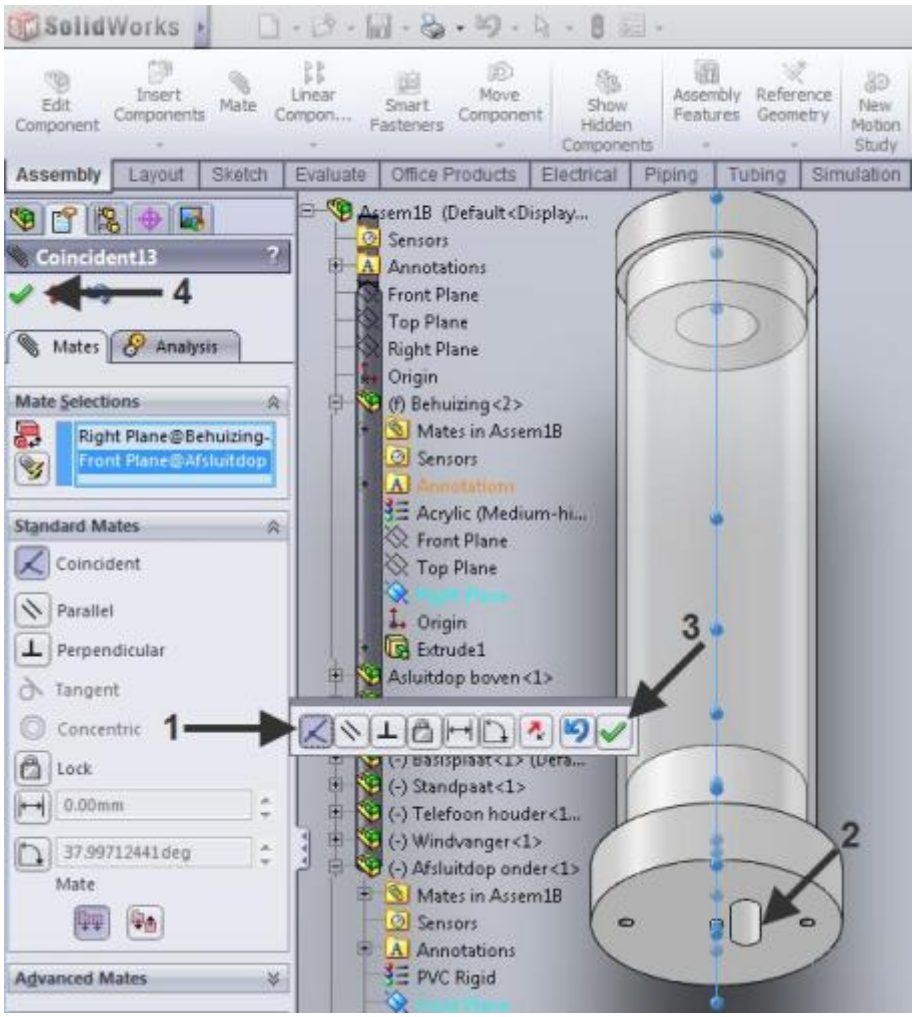
1. As 'Mate', SolidWorks has selected 

.
2. The keyway must be on the right side. If not, read the instructions below.
3. Click OK to attach the 'Mate'.

4. Click OK once more to correctly close the Property Manager.

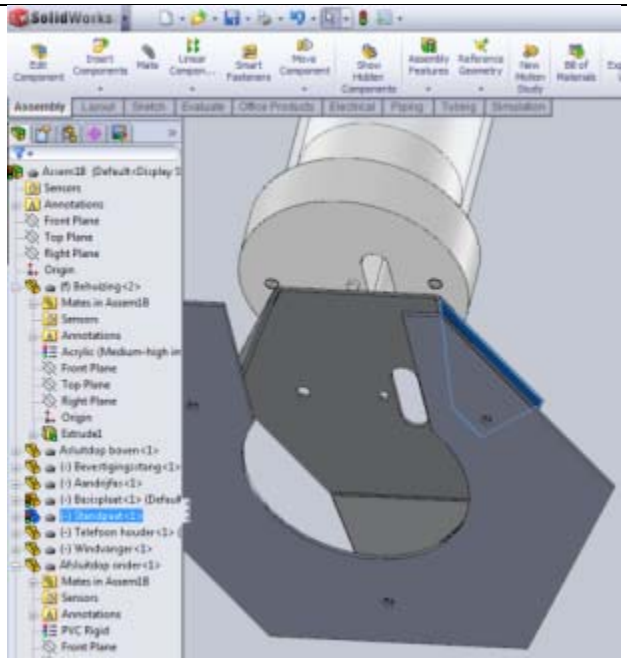
To get the keyway on the right side, you might use

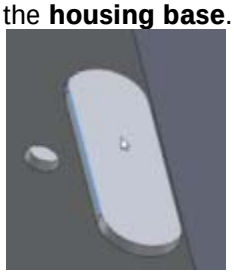
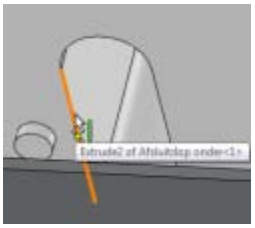
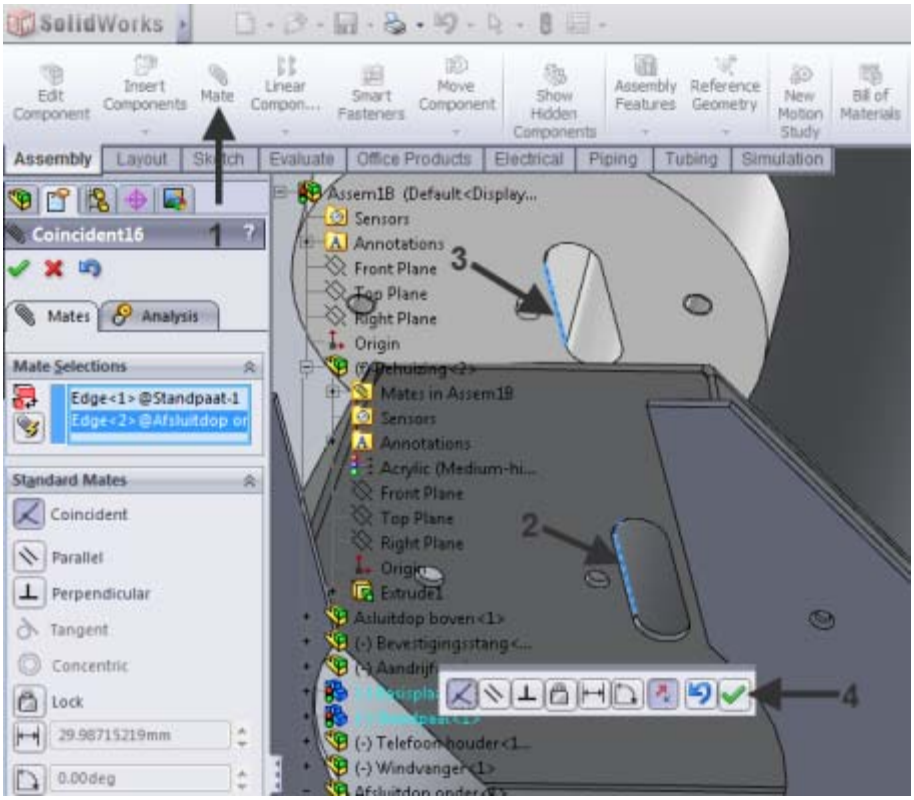
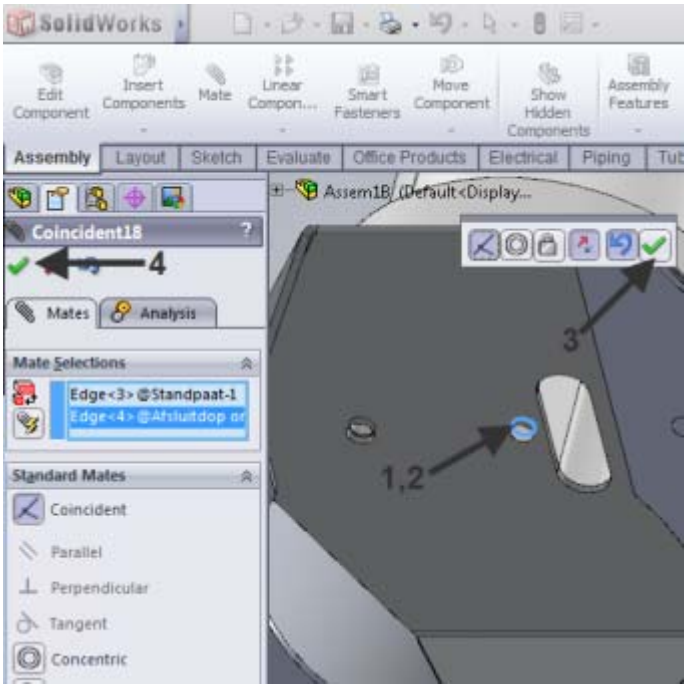


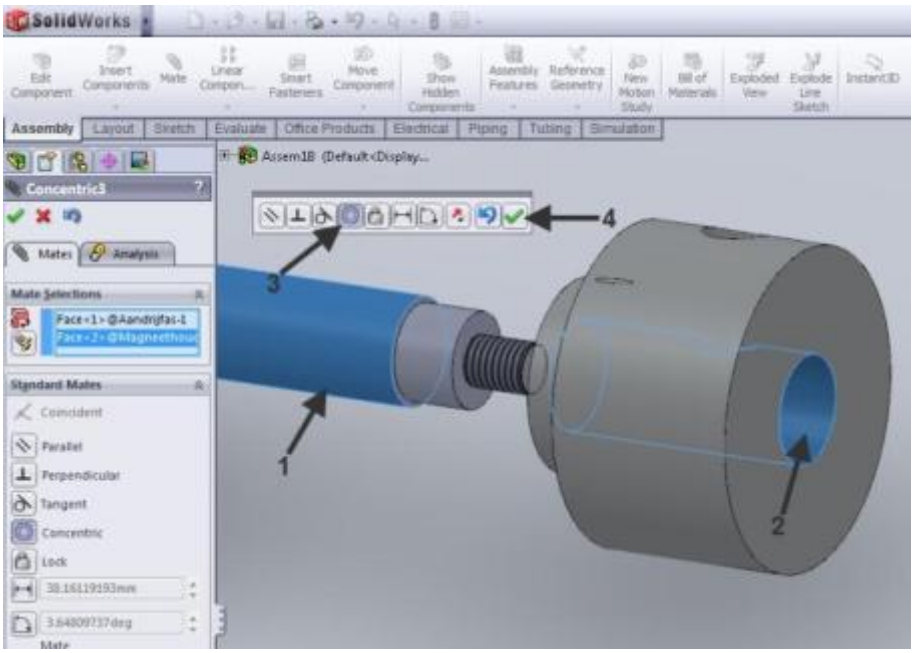
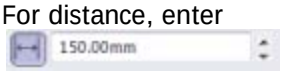
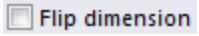
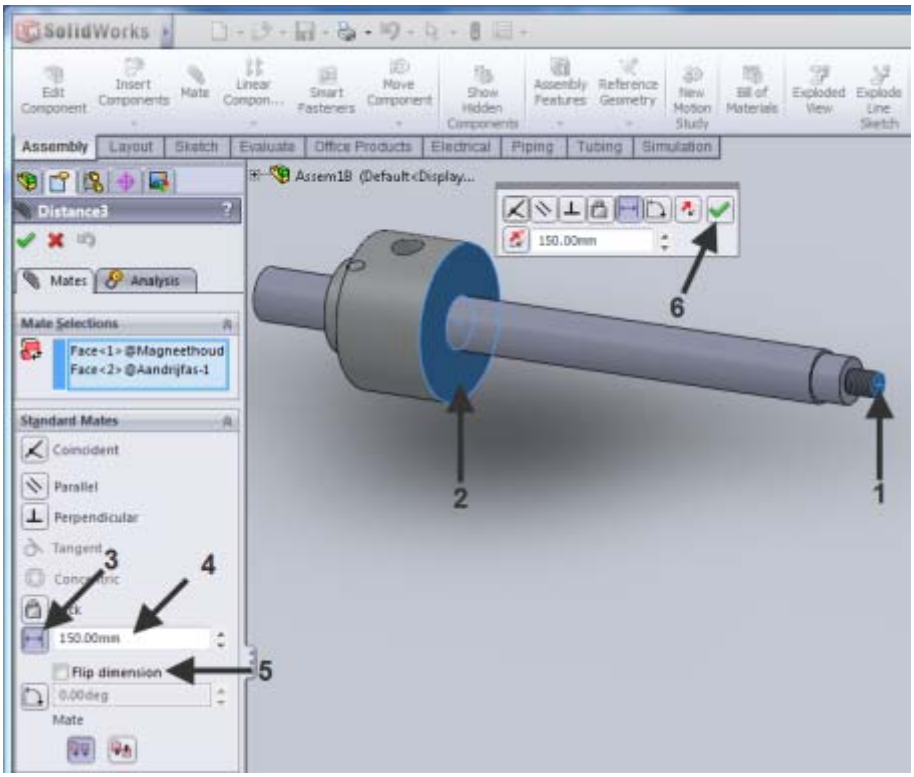


27

Zoom in, drag and/or rotate the bearing plate and the base sheet as illustrated.
Both are linked together, which is why they move together!



28	Now, first Mate both keyways together.  **1.** Click **2.** Click the upper edge of the **housing base**.  **3.** Next, select the lower edge of the **bottom end**.  **4.** Confirm this by clicking OK.	
29	**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.	

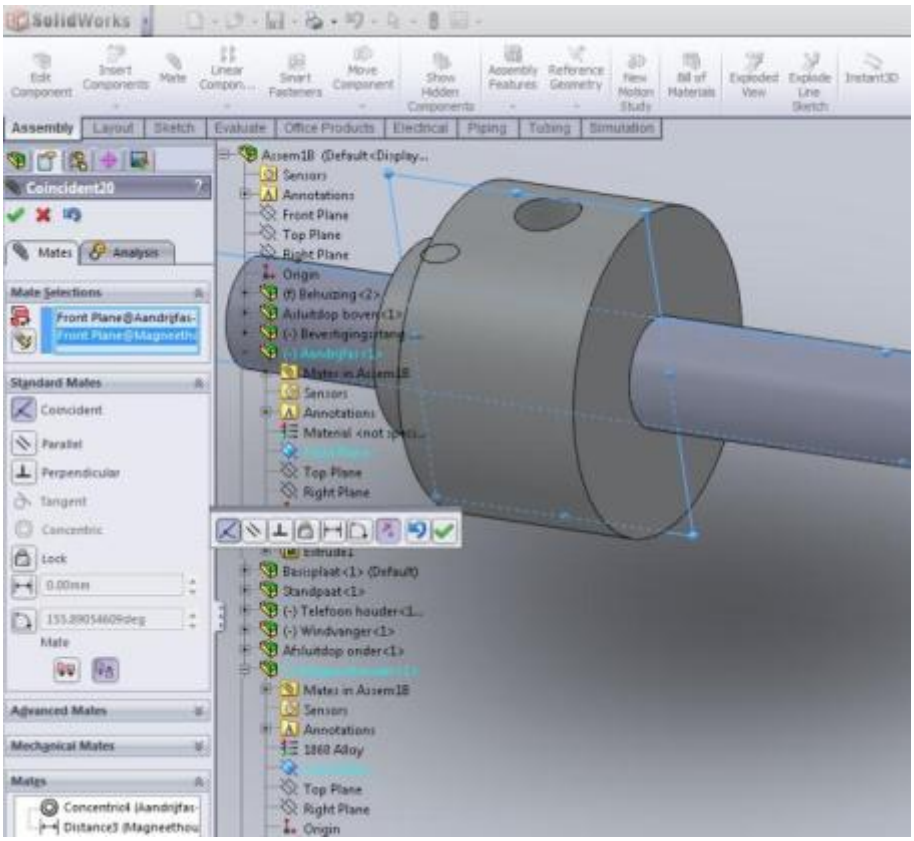
30	Zoom in, drag and/or rotate the **shaft** and the **magnet holder** as illustrated. Next click:  to link both pieces. **1.** Select the outer face of the **shaft.** **2.** Then choose the inner face of the hole. **3.** Using the function:  link both pieces together. **4.** Click OK: 	
31	**1.** Select the upper face of the **shaft.** **2.** Then choose the upper face of the **magnet holder.** **3.** For mate, select distance  . **4.** For distance, enter  **5.** Check or uncheck  to move the **magnet holder** to the right place. **6.** Click OK.	

32

now, the:

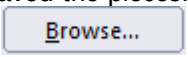
of the **shaft** and the **magnet holder.**

If you don't know how to do that, review steps 8 through 13 or 25 through 26.

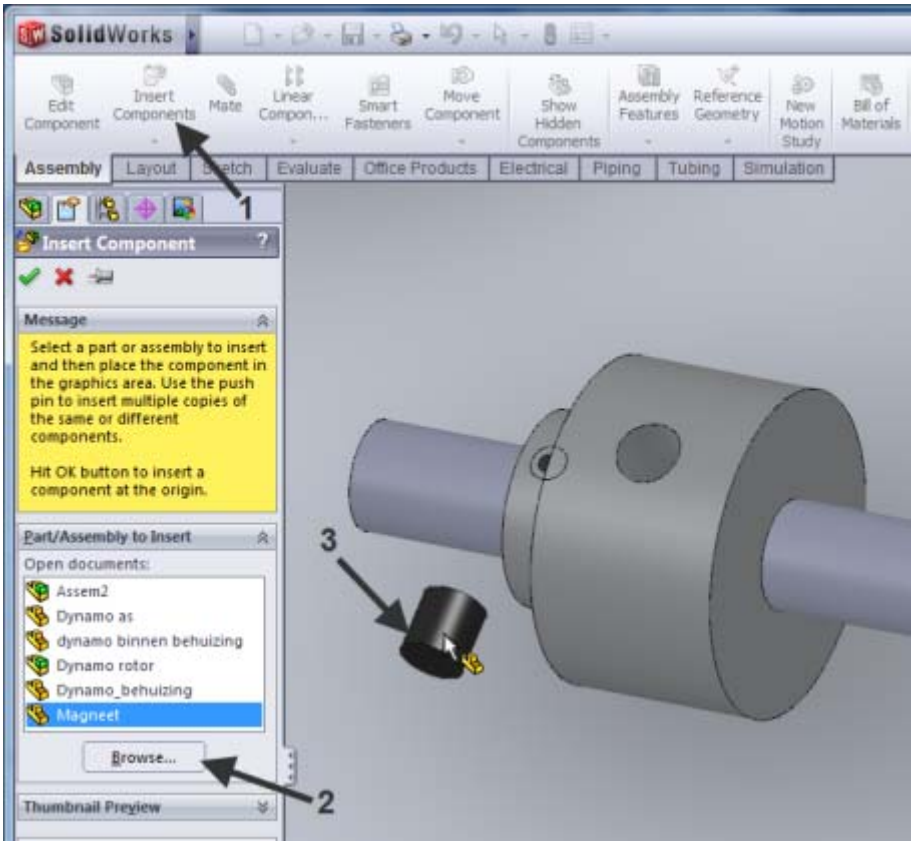


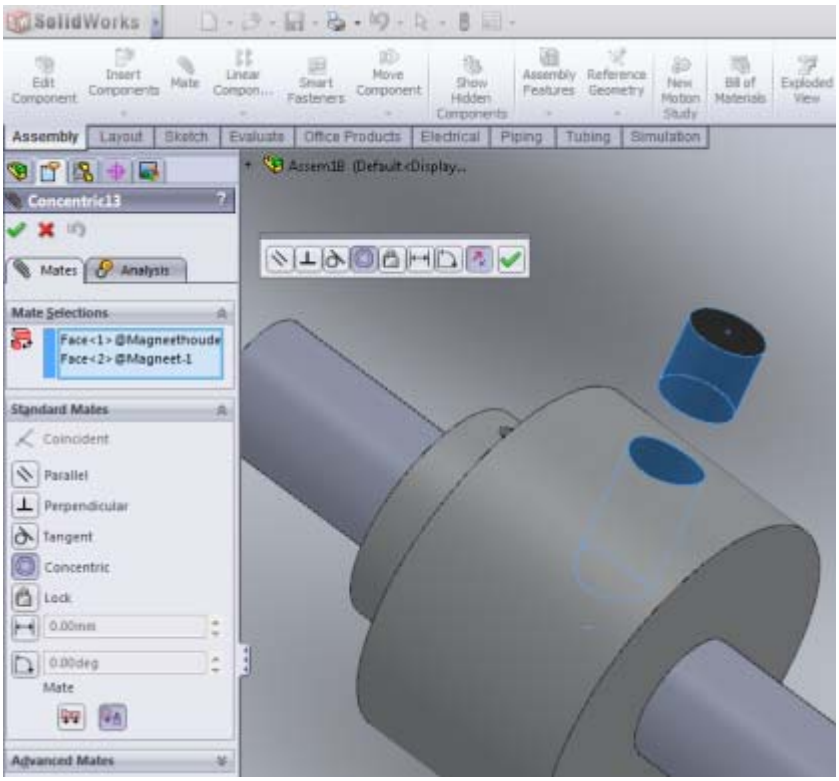
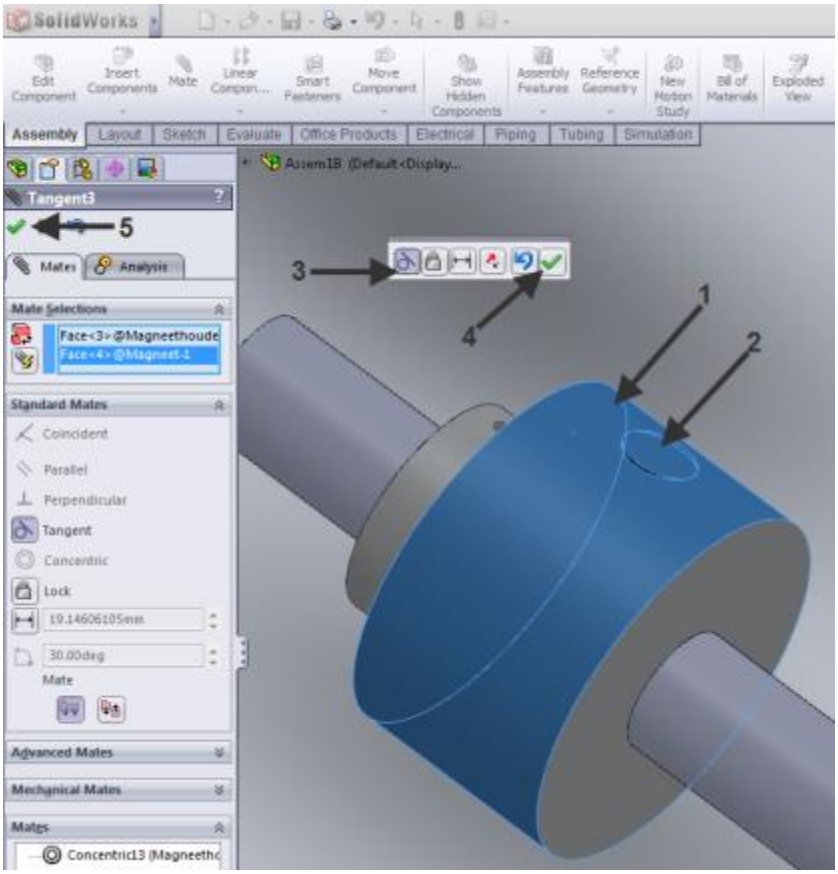
33

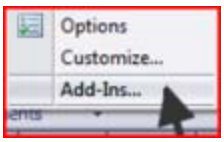
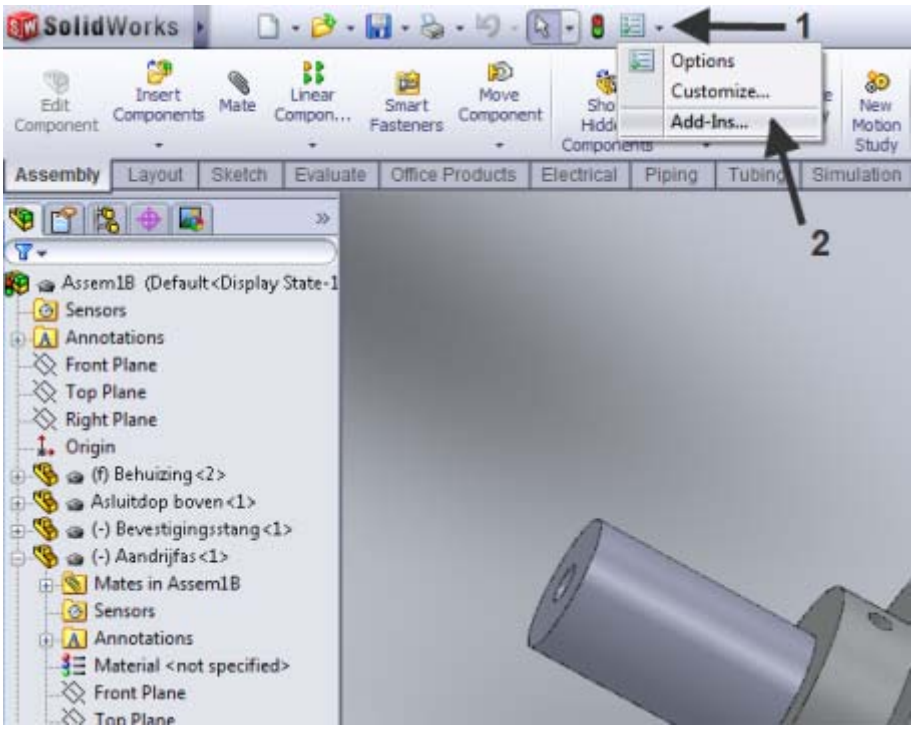
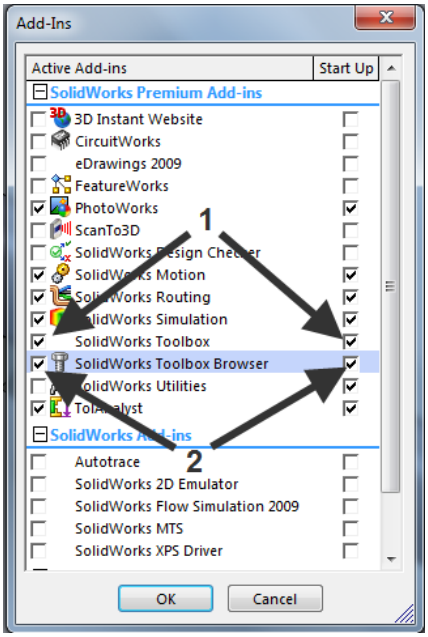
To finalize the piece, you must position two more pieces.
You need a **magnet** and an M6x8mm Allen head bolt.

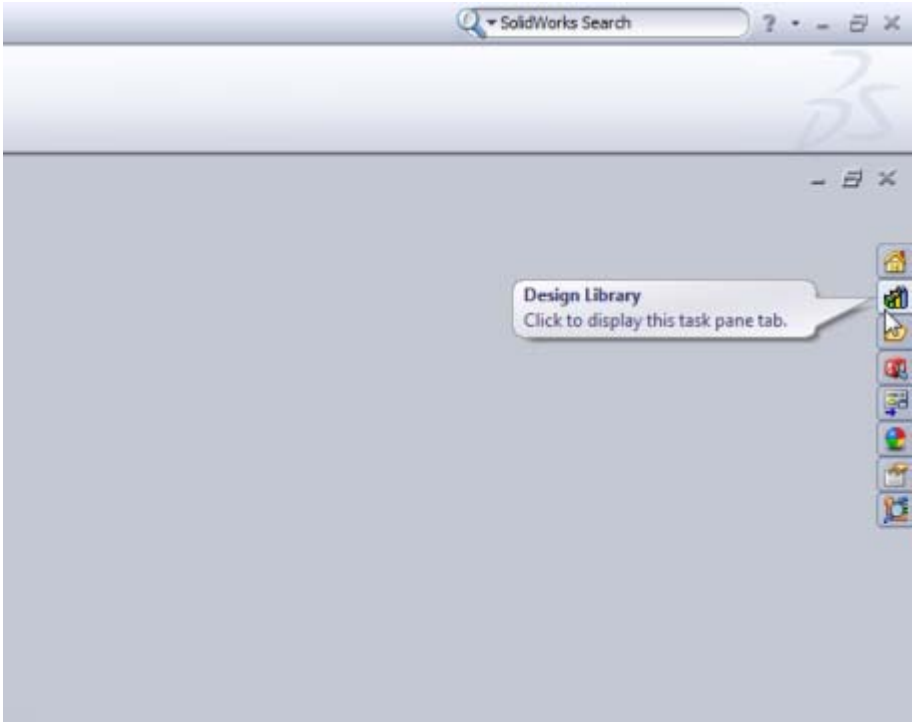
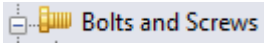
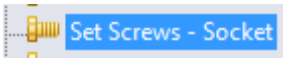
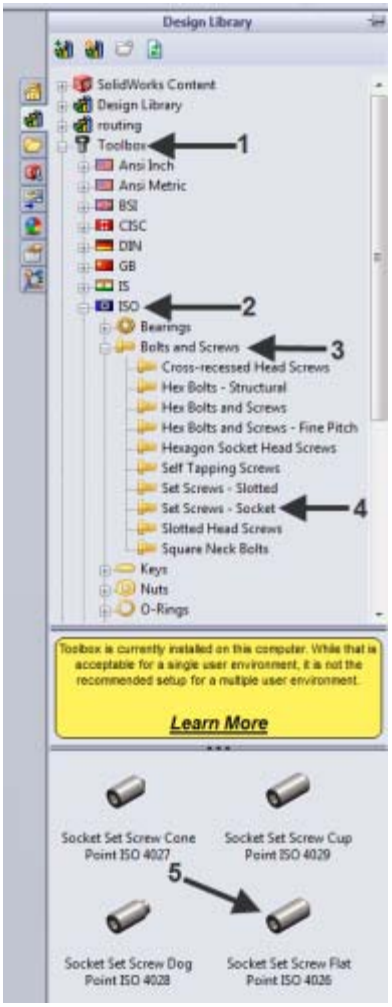
1. Click:
2. Go to the folder where you saved the pieces.
   Click: 

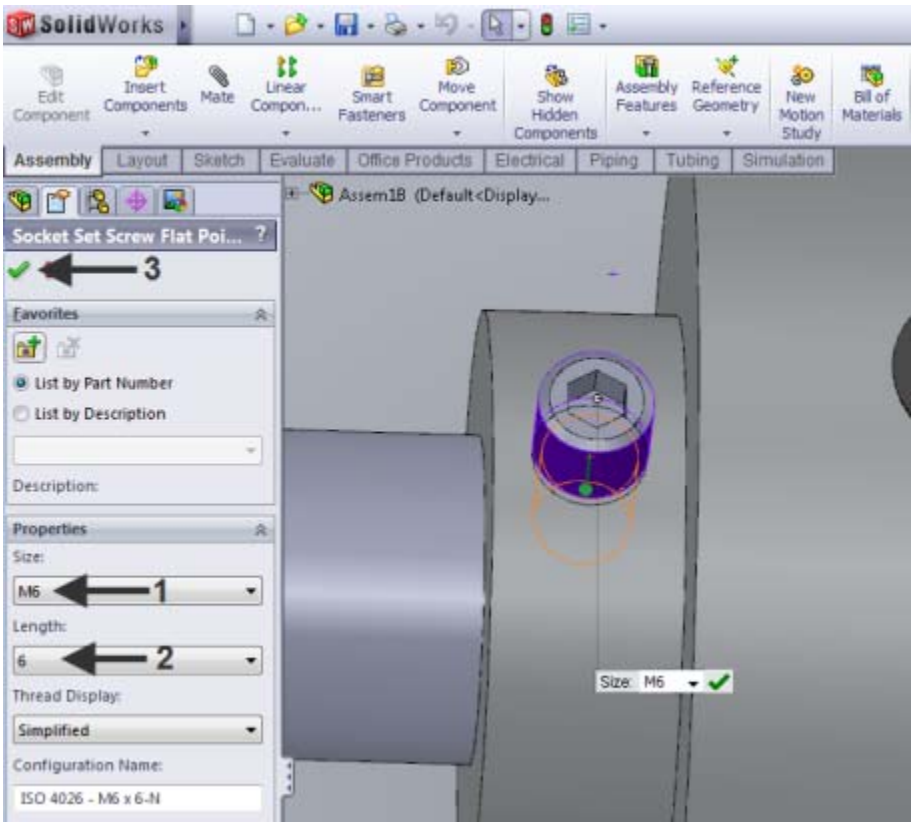
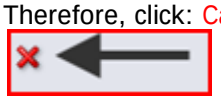
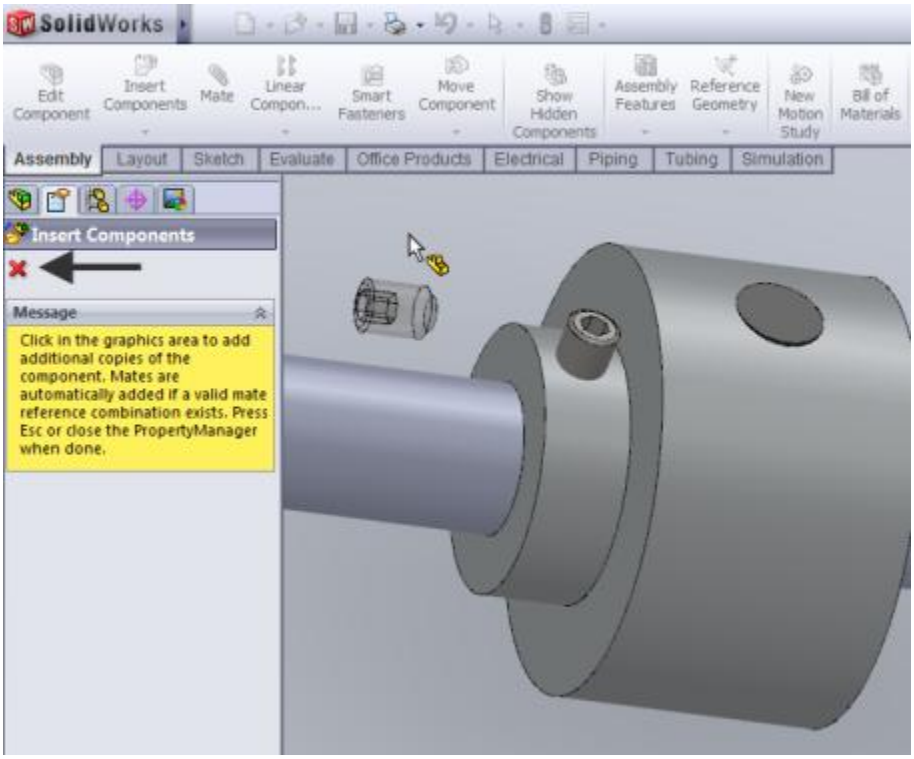
3. Add the **magnet**.

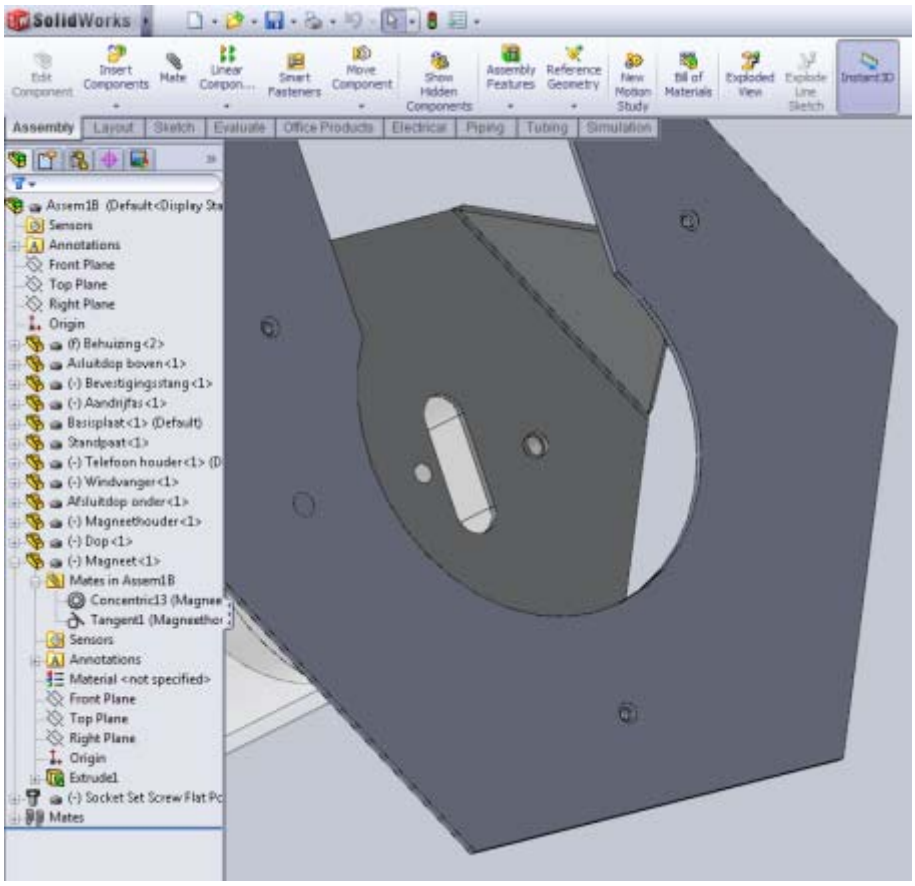


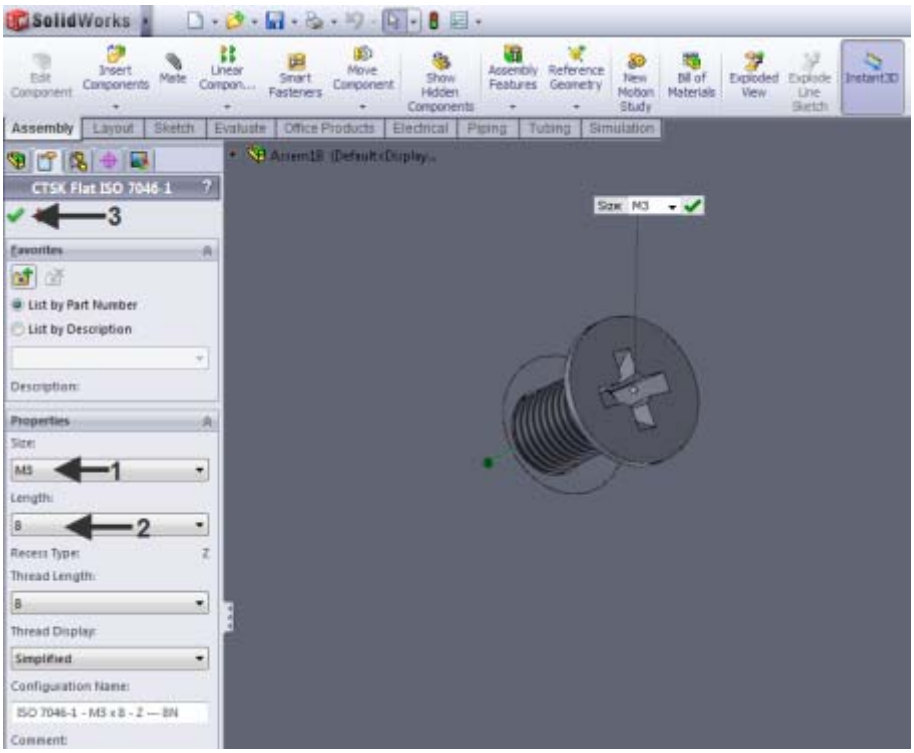
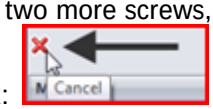
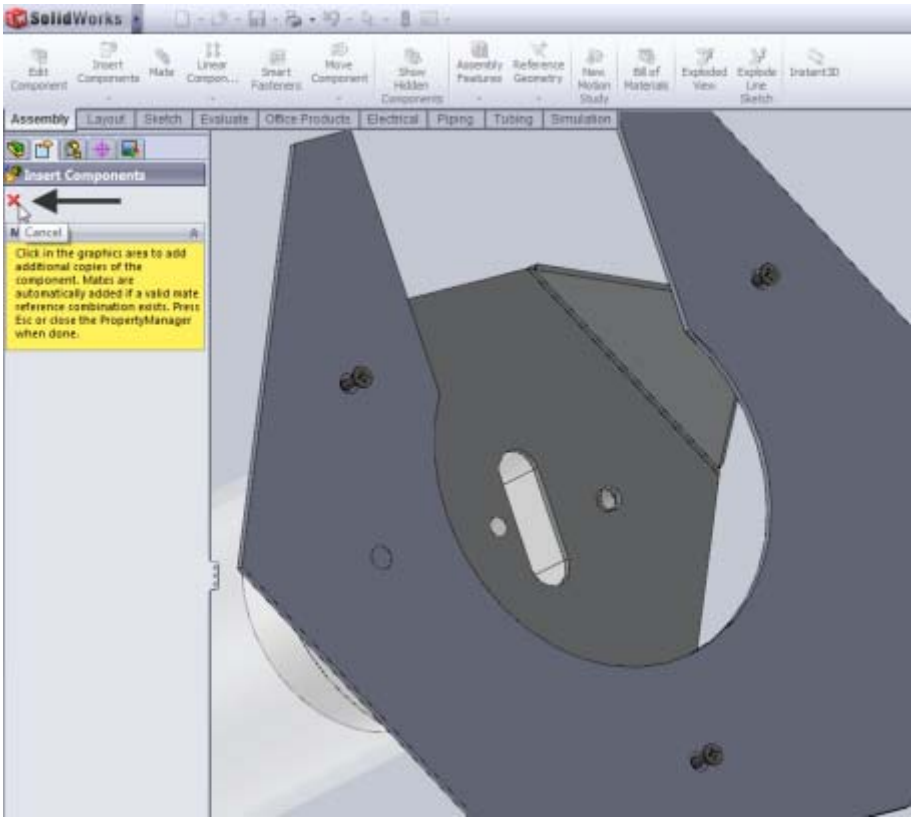
34	 now the outer face of the **magnet** with the inner side of the hole. Then click OK:	
35	**1.** First, select the outer face of the **magnet holder.** **2.** Then select the upper side of the **magnet.** **3.** In this case, SolidWorks has selected a:  mate. **4.** Click OK to confirm the mate.  **5.** Click OK once more.	

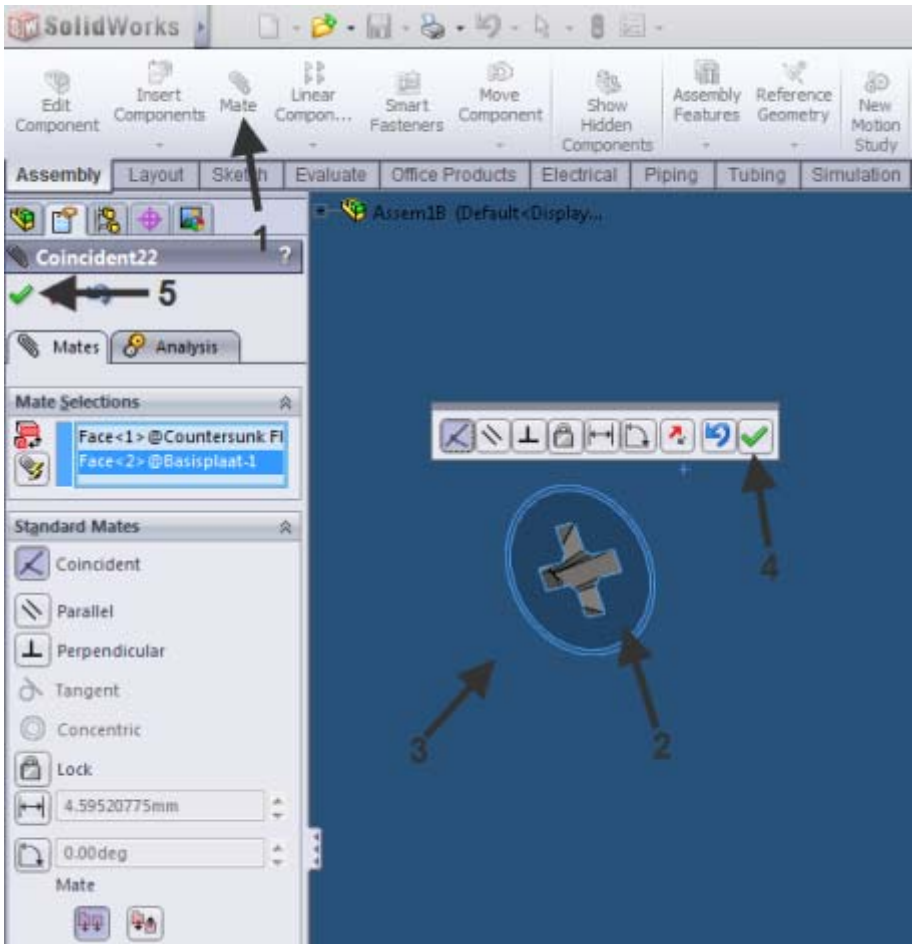
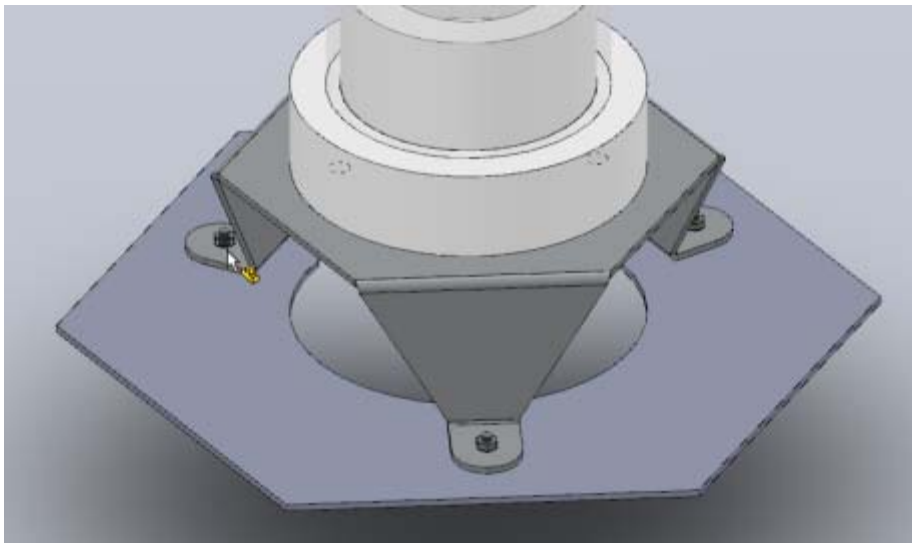
36	We will now add an M6x8mm Allen head bolt. We will do that using the SolidWorks Toolbox function. Before continuing, you must first make sure Toolbox has been installed and activated on your computer. **1.** In the Command Manager, click the arrow next to  **2.** Then choose: 	
37	Make sure the SolidWorks Toolbox and SolidWorks Toolbox Browser options are both checked in the menu. By adding a check mark to the right, **after** both options (SolidWorks Toolbox and SolidWorks Toolbox Browser), they will be, from now on, automatically loaded when SolidWorks is launched. So you don't have to activate the Toolbox each and every time. Read the following tip in case these options are unavailable.	
		It may be Toolbox is not available in your version of SolidWorks. In that case, you will not be able to finish the tutorial by following the steps below. If you still want to complete the model, you can also download the required attachments (bolts and washers) from www.solidworks.nl. You will not be using the Toolbox, but will add the bolts and washers to the assembly, as you would do with any other piece.

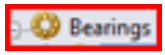
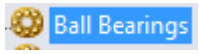
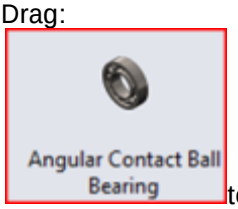
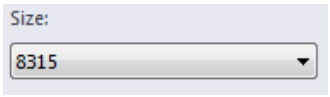
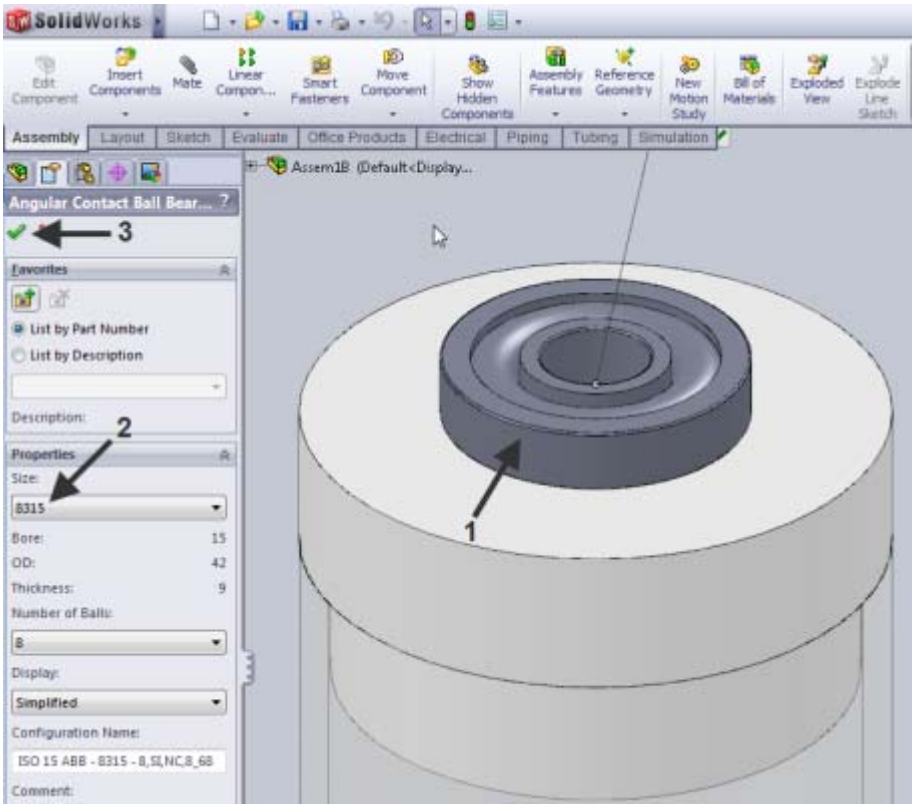
38	In the task pane Task Pane: (to the right on the screen), click the Design Library icon. 	
39	The Task Pane will open with the Toolbox. We will now insert an Allen head bolt into the threaded hole. Successively double-click **1.**  **2.**  **3.**  **4.**  The available screws will be displayed in the lower part of the Task Pane. **5.** Search for the screw with the following name: 	

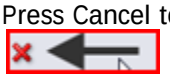
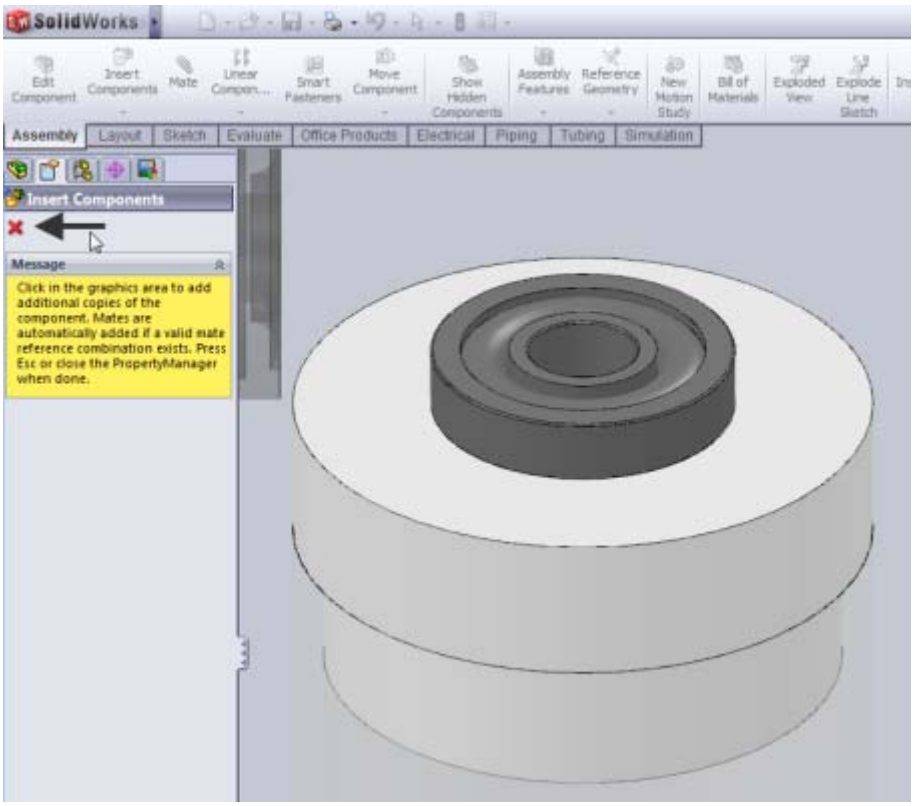
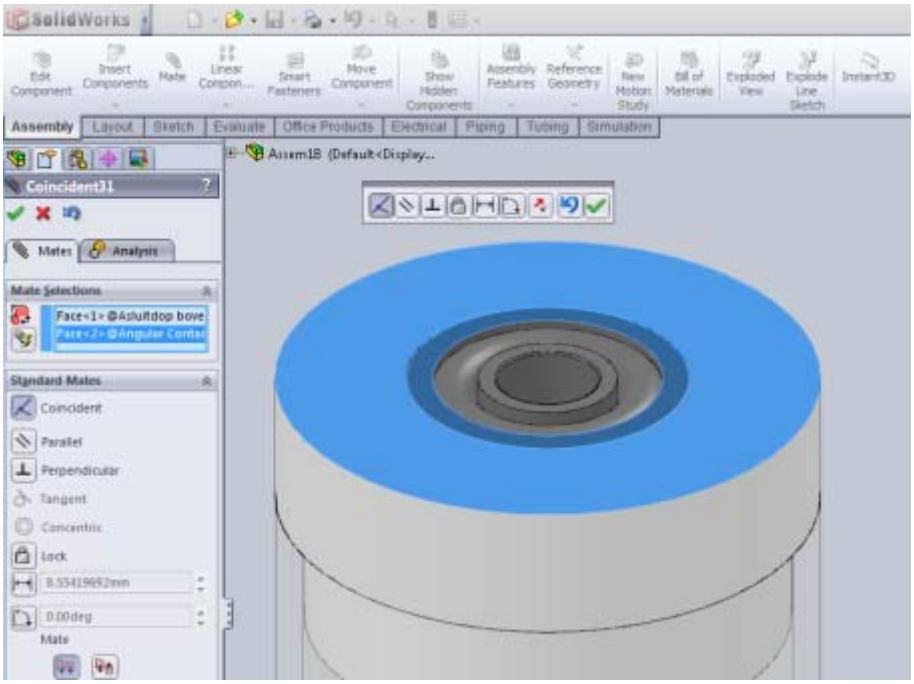
40	With the left mouse button, drag the screw from the Task Pane to your model. As soon as the mouse moves above the threaded hole, the screw jumps to the appropriate position. Release the mouse button. The screw may seem much too small or too large. That is not important at this point. In the: Property Manager change the size of the screw to M6x8, and click OK.	
41	The screw is now locked to the mouse and you could insert it into other threaded holes. But because we don't have any other holes, we no longer need the screw. Therefore, click: Cancel 	

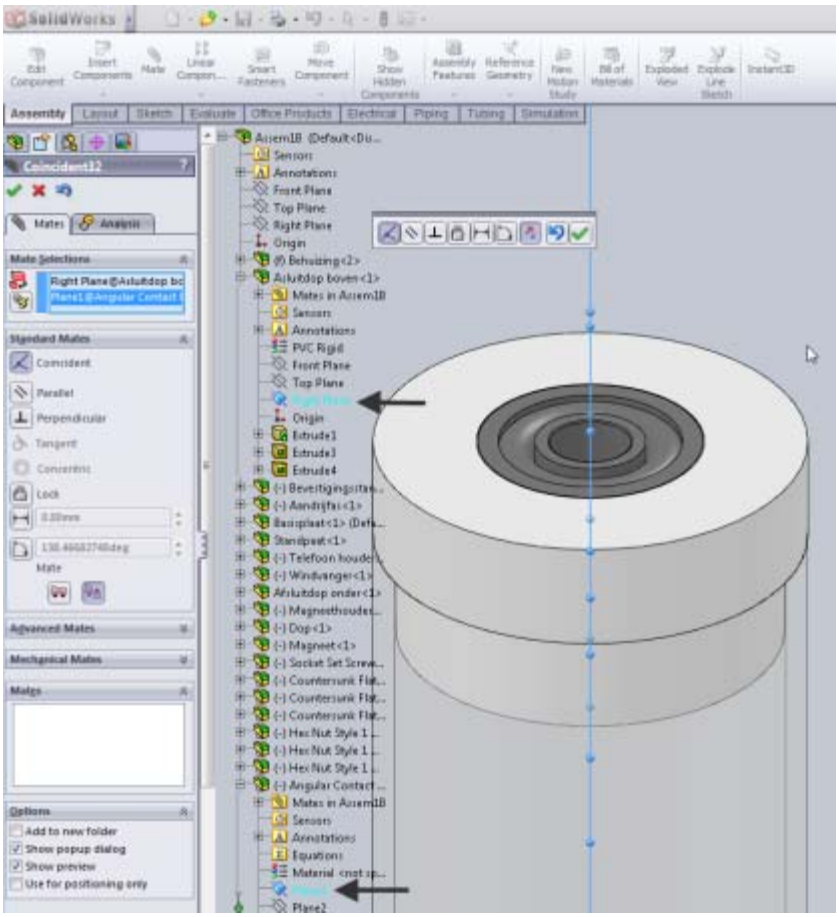
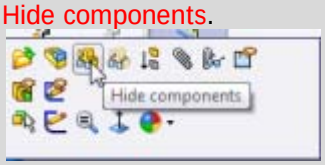
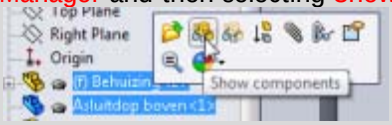
42	Zoom in, drag and/or rotate the **<u>housing base</u>** and the **<u>base sheet</u>** as illustrated.	
43	Add the following piece in the same way as above. Successively double-click **1.** Toolbox **2.** ISO **3.** Bolts and Screws **4.** Cross-recessed Head Screws The available screws will be displayed in the lower part of the Task Pane. **5.** Search for the screw with the following name: 	

44	Drag this screw to the hole. In: Property Manager,, 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: 	

46	It may be the screws are protruding. Solve that problem as follows.  **1.** Click: **2.** Select the upper face of the screw. **3.** Then select the face of the base sheet. **4.** Click OK: **5.** Click OK once more: Repeat this for both other screws.	
47	Add the following piece to the bolt ends in the same manner. (M3 Bolt)  	

48	We will add a **bearing**. You will use the toolbox once more. Double-click the following pieces. **1.**  **2.**  **3.**  **4.** Choose  **5.** Drag:  to the hole of the **cap internal.**	
49	**1.** Insert the **bearing** into the hole. **2.** Locate the appropriate bearing  **3.** Click OK:	

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

52	Then mate:  of **the bearing** and the  of the **cap internal**. This is required to fasten the bearing.	
		Sometimes a piece is in the way during the assembly. For instance, it may not be possible to select a piece correctly. This can be solved in two ways. You can hide the piece that is in the way by clicking it and then selecting Hide components.  You can bring it back again by clicking the hidden piece in the Feature Manager and then selecting Show components.  Or, you can make the piece transparent. Again, click the piece and then select Change Transparency.  If you want the piece to be displayed normally again, click it once more and then click again Change Transparency.

53

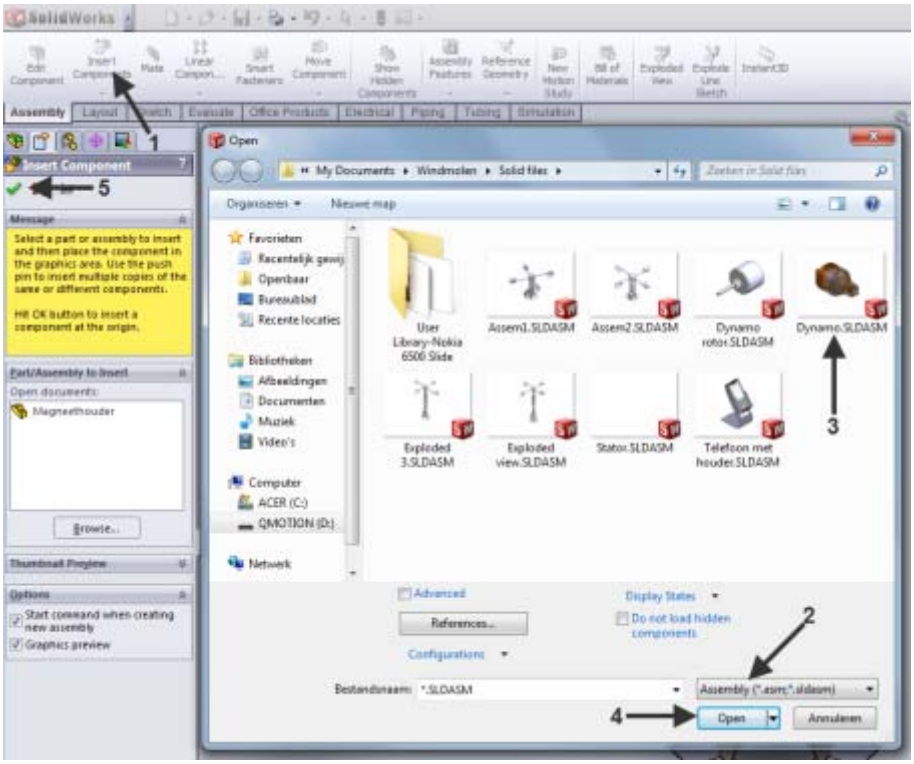
Add the **Dynamo** to the assembly.

1. Click: 

2. Change the search to:
Assembly (*.asm;*.sldasm)

3. Then choose the file:


4. Click: 



54

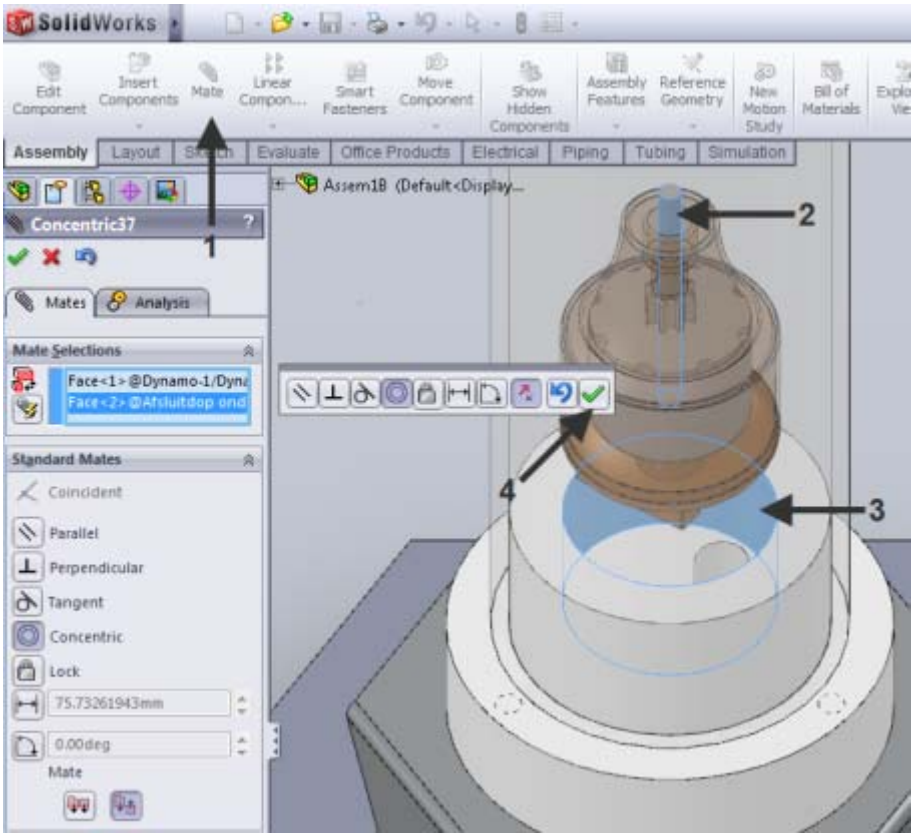
1. Click: 

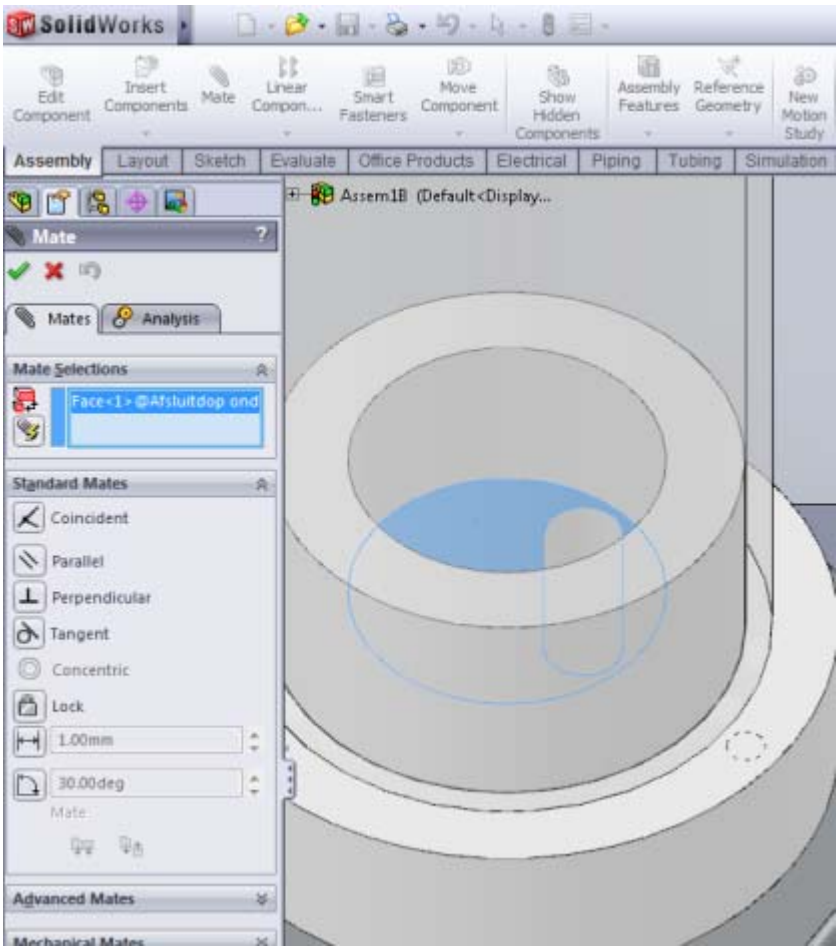
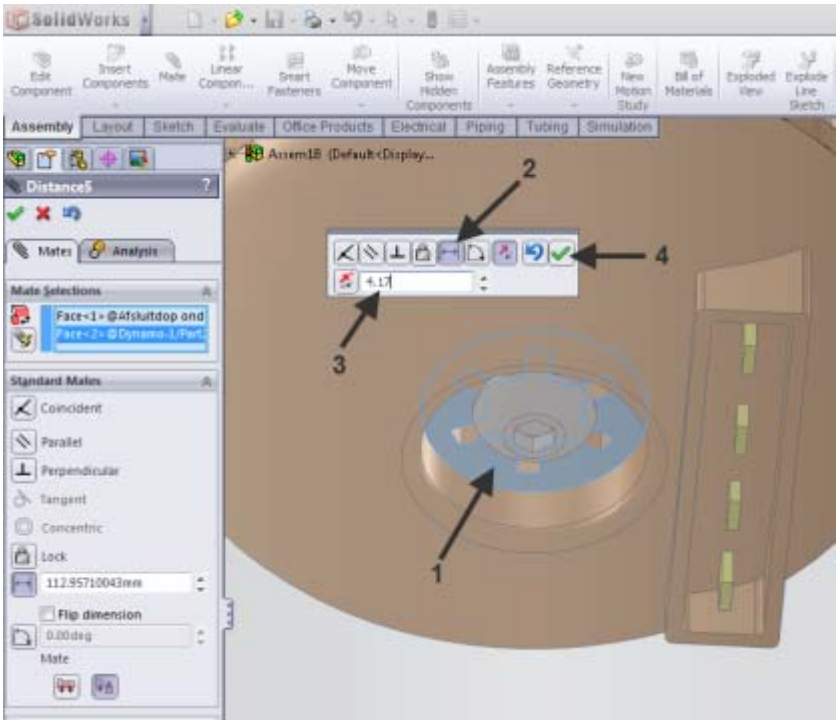
2. Choose the outer edge of the shaft.

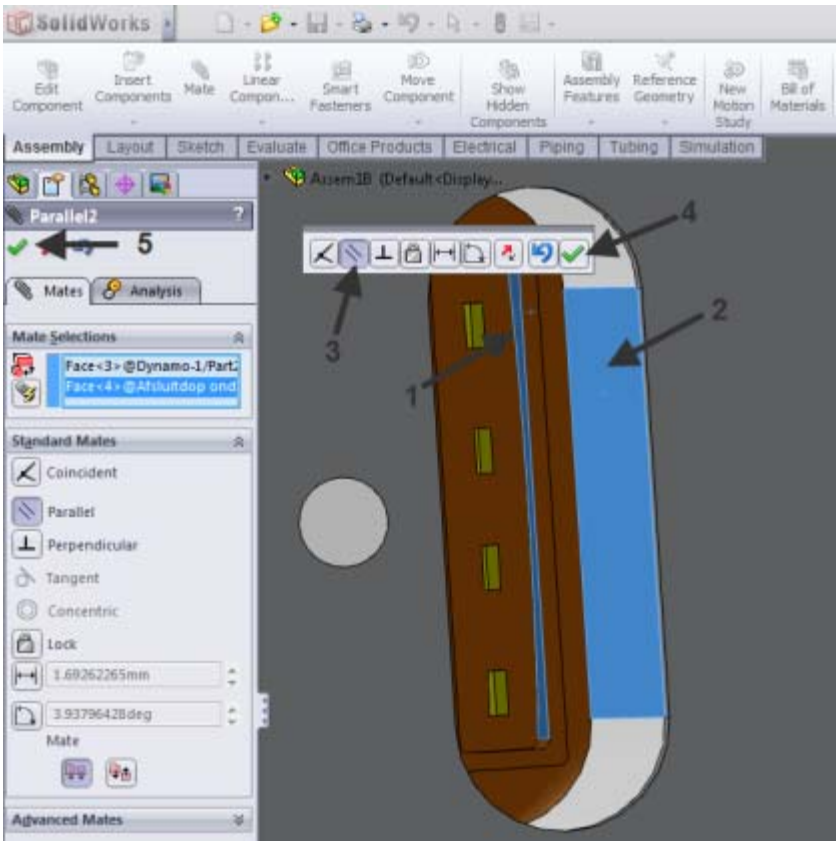
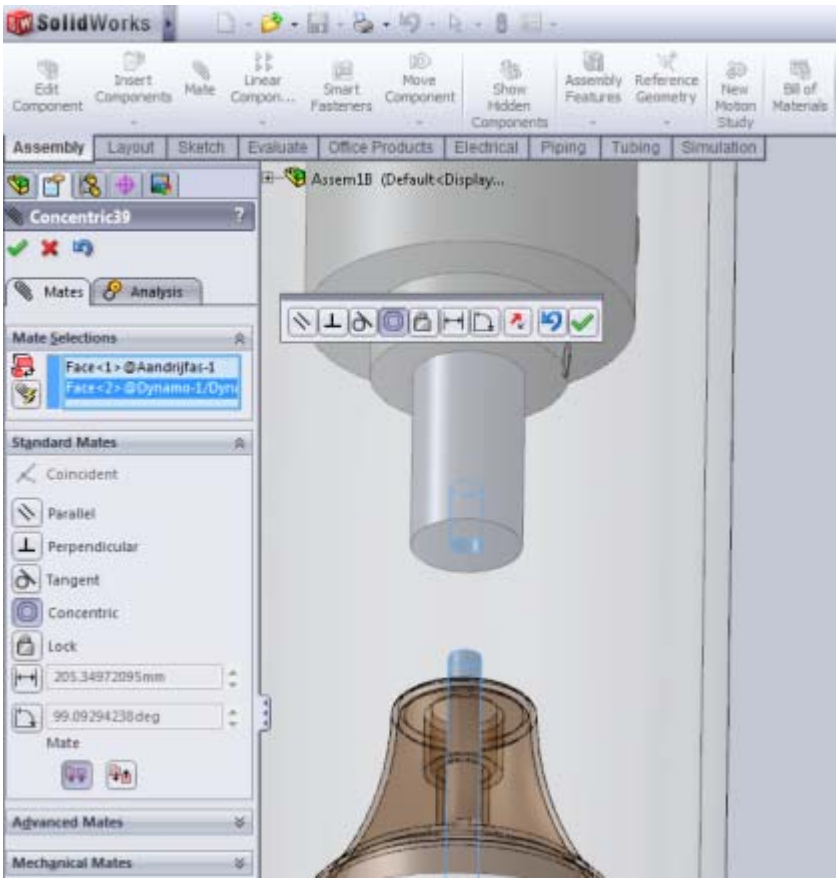
3. Then choose the interior of the **bottom end.**

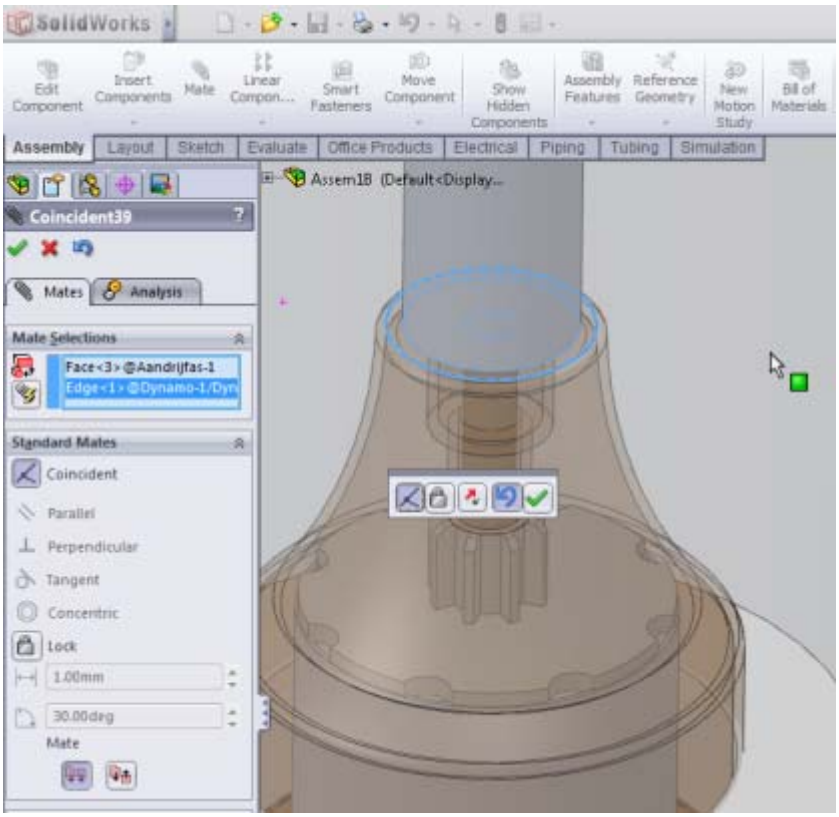
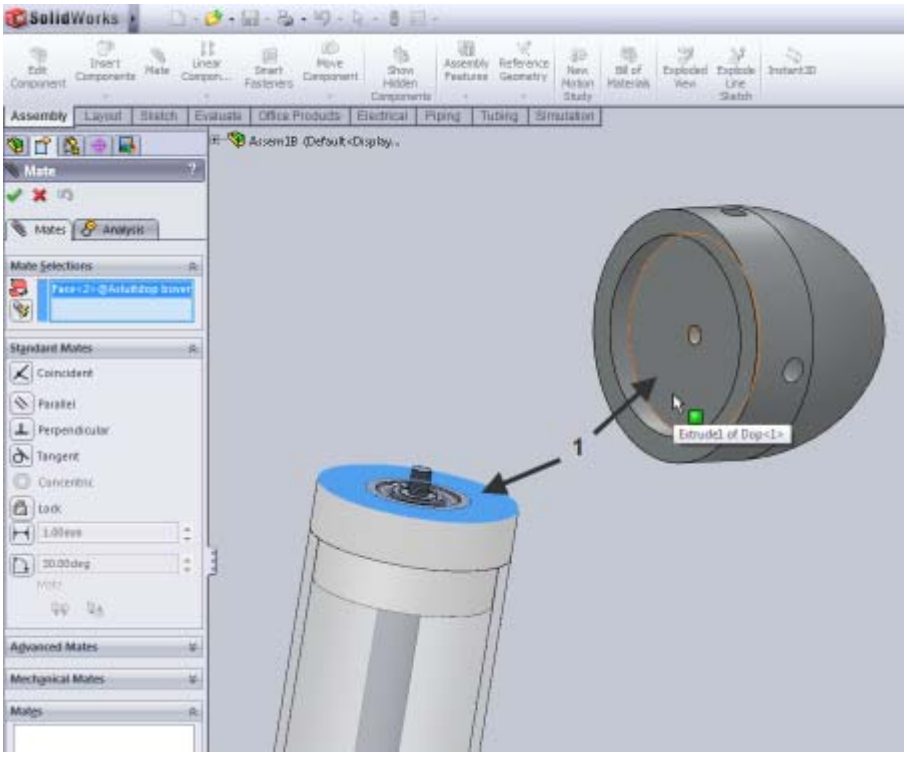
4. Click OK:

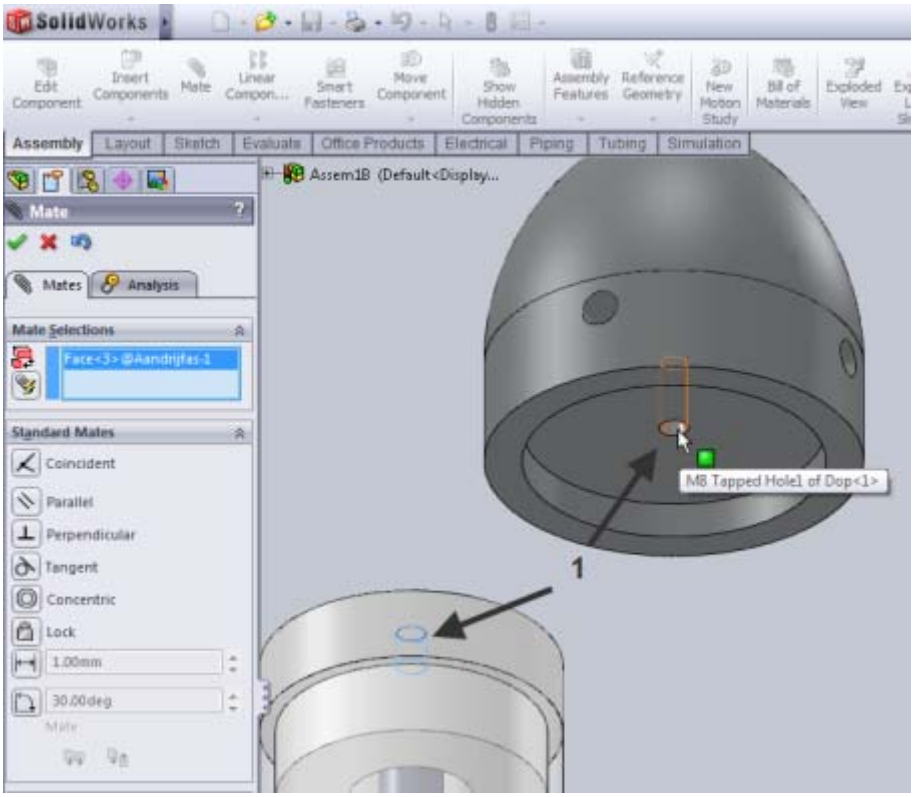
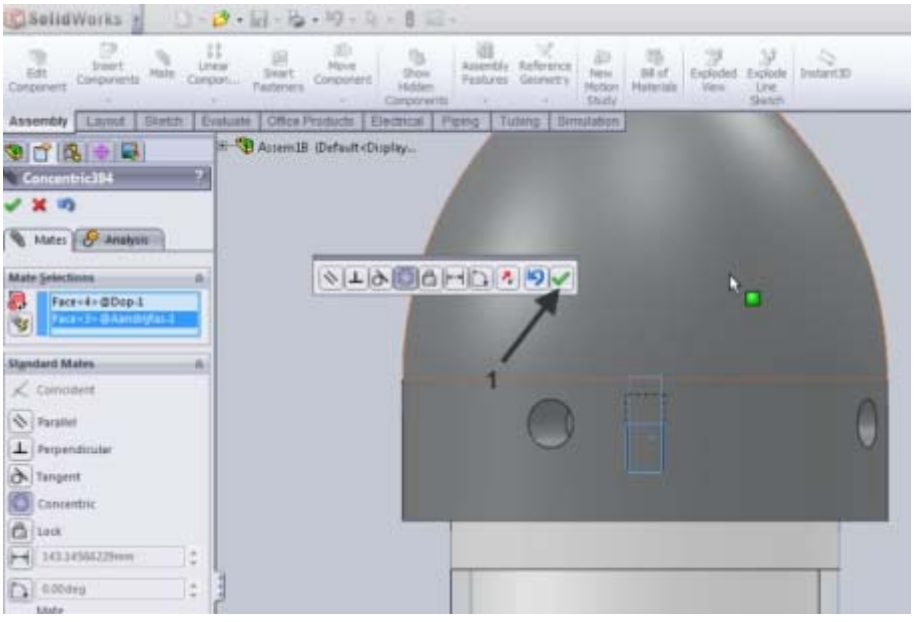

5. Click OK once more:

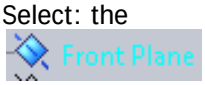
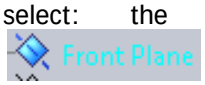
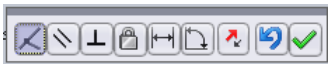
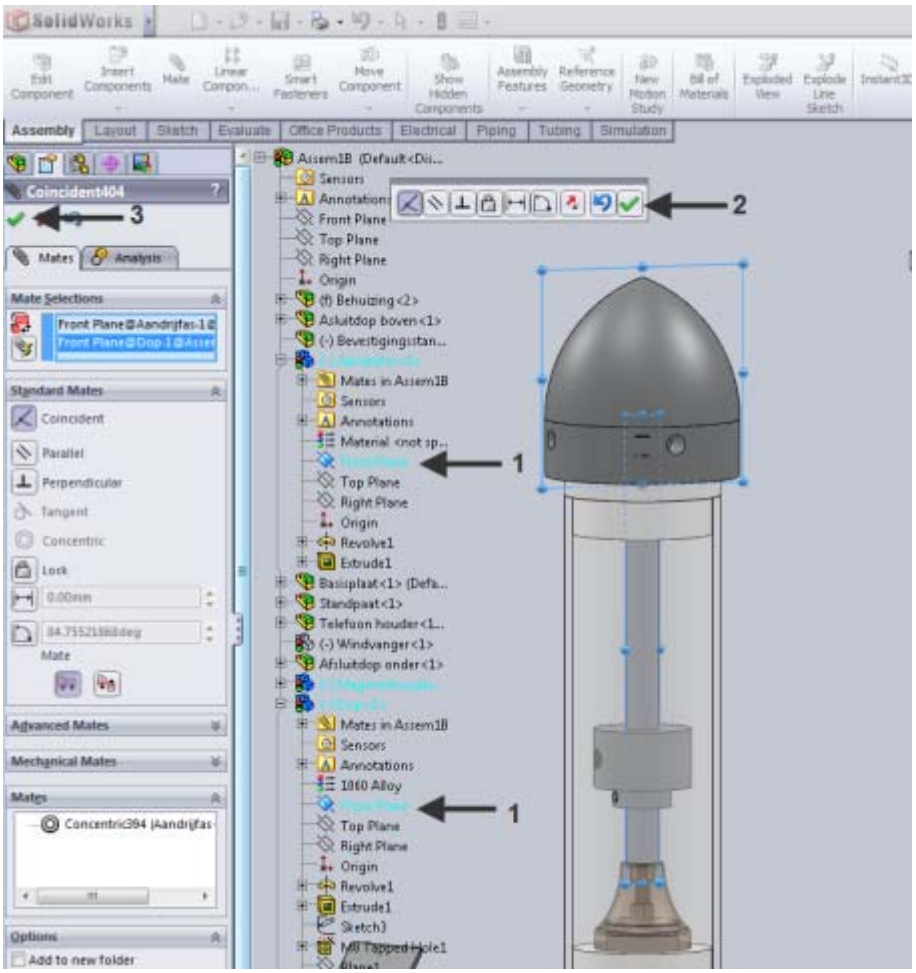
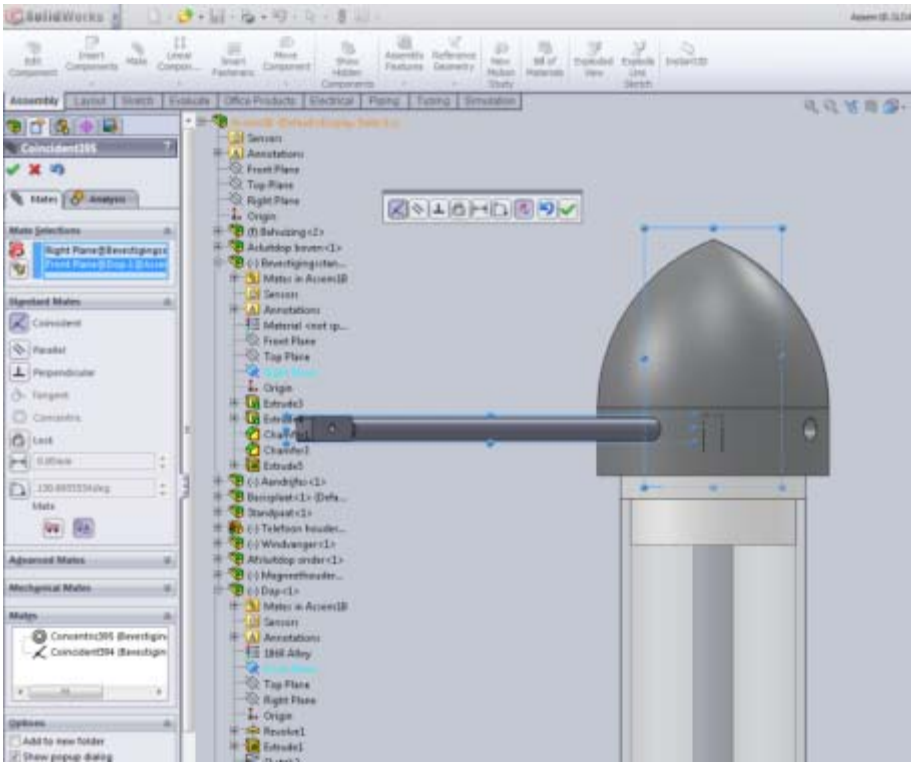


55	Select the bottom of the hole of the: **bottom end.**	
56	**1.** Select the bottom of the hole of the dynamo. **2.** Then select the mate:  **3.** Enter 4.17mm:  **4.** Click OK: 	

57	**1.2**. Select the faces as in the figure: **3.** For mate enter:  Parallel **4.** Click OK:  **5.** Click OK once more:	
58	Now mate the shaft of the dynamo to the hole of the shaft. See the figure!	

59	Complete the mate by linking the bottom of the shaft and the top of the dynamo.	
60	Now is the time to put the **cap** to place. **1.** Click mate and select the upper face of: **cap internal** and the inner face of **top end.** Mate these two pieces together.	

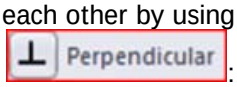
61	**1.** Select the threading of the shaft and the threaded screw hole M8 of the cap.	
62	Click OK once.	

63	**1.** Select: the  of the **<u>shaft,</u>** Then select: the  of the **<u>top end.</u>** **2.** Click OK.  **3.** Click OK once more:	
64	Mount the wing arm to the top end. Use the following mates to do that. Concentric for the shaft/hole assembly. Coincident for the end shaft / end hole mount. Finally, use Planes to straighten the wing arm. See figure.	

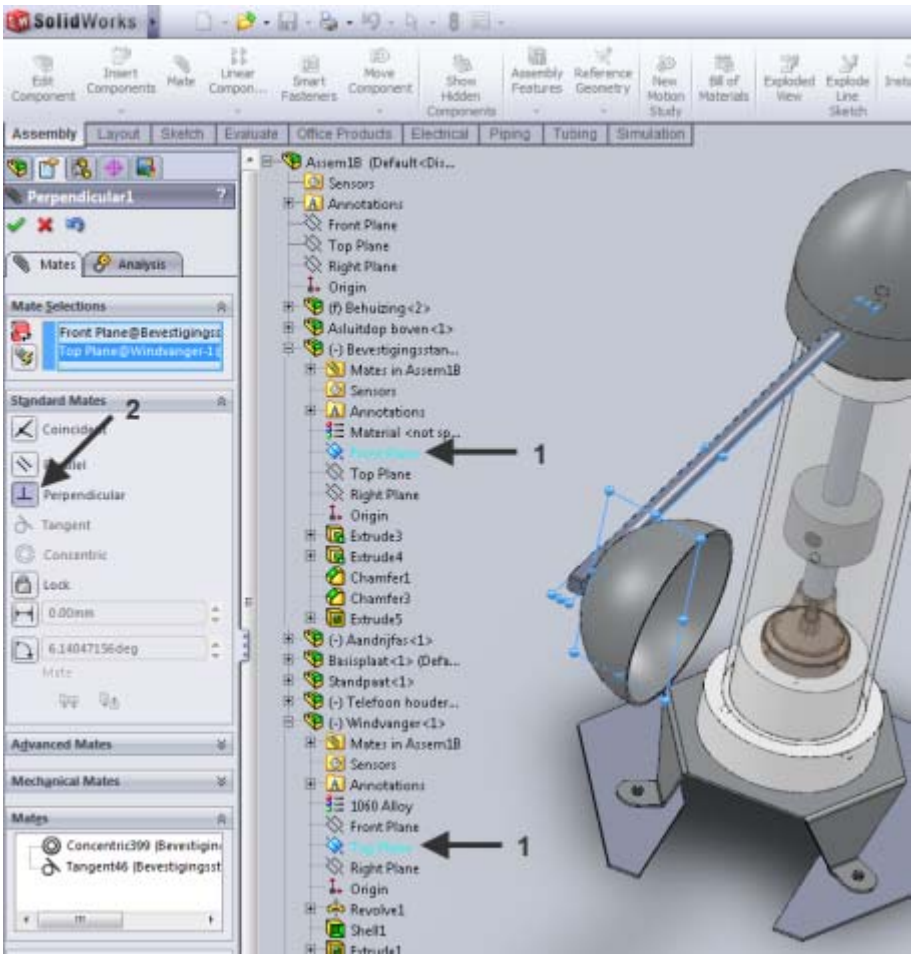
65

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.

1. Finally, choose the Front Plane of the wing arm and the Top plane of the windblade.
2. Put them straight to each other by using 

:

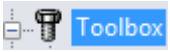


66

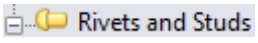
Get the piece:

from the Toolbox.

Double-click the following pieces

1. 

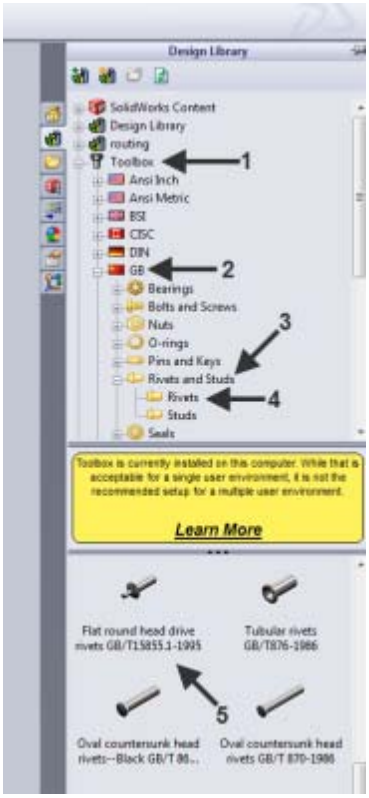
2. 

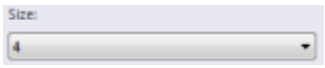
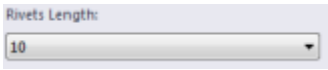
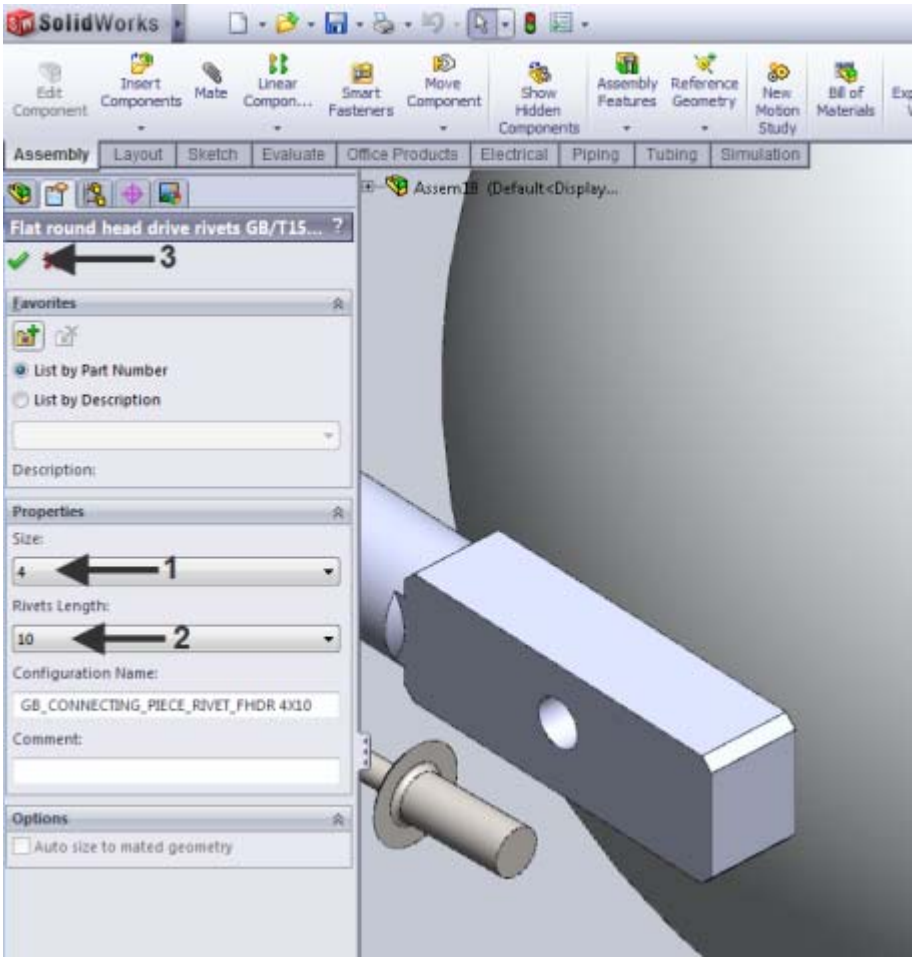
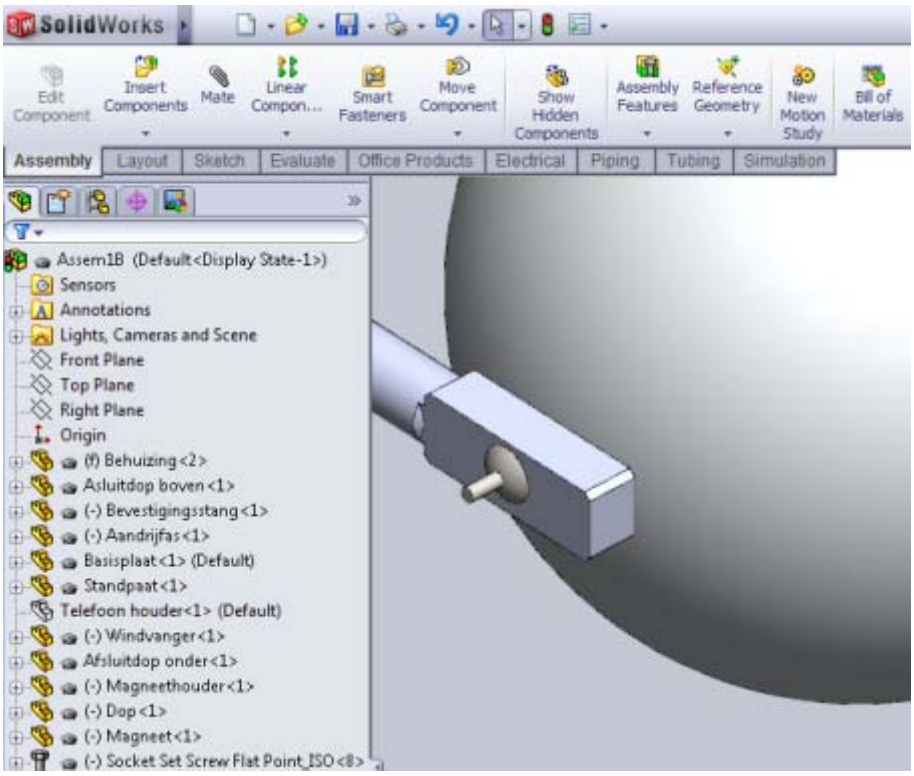
3. 

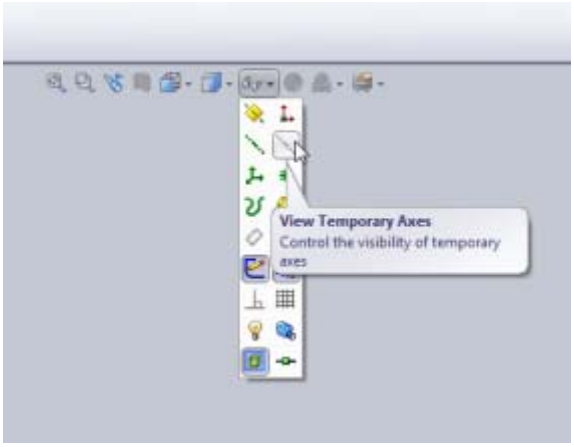
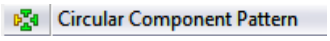
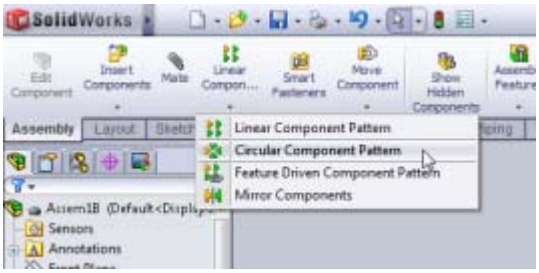
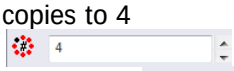
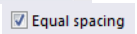
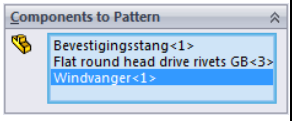
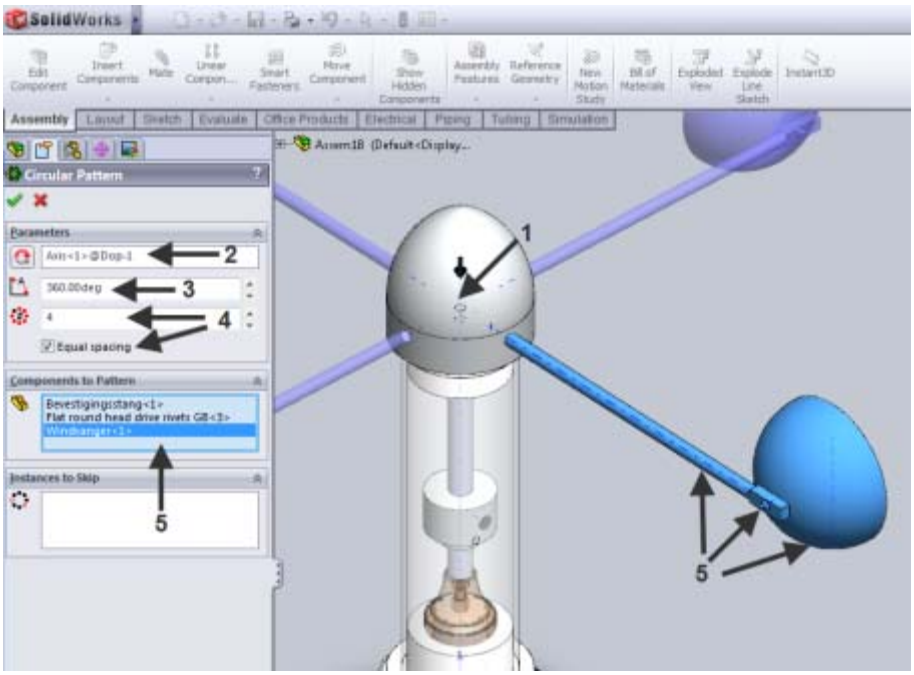
4. Choose 

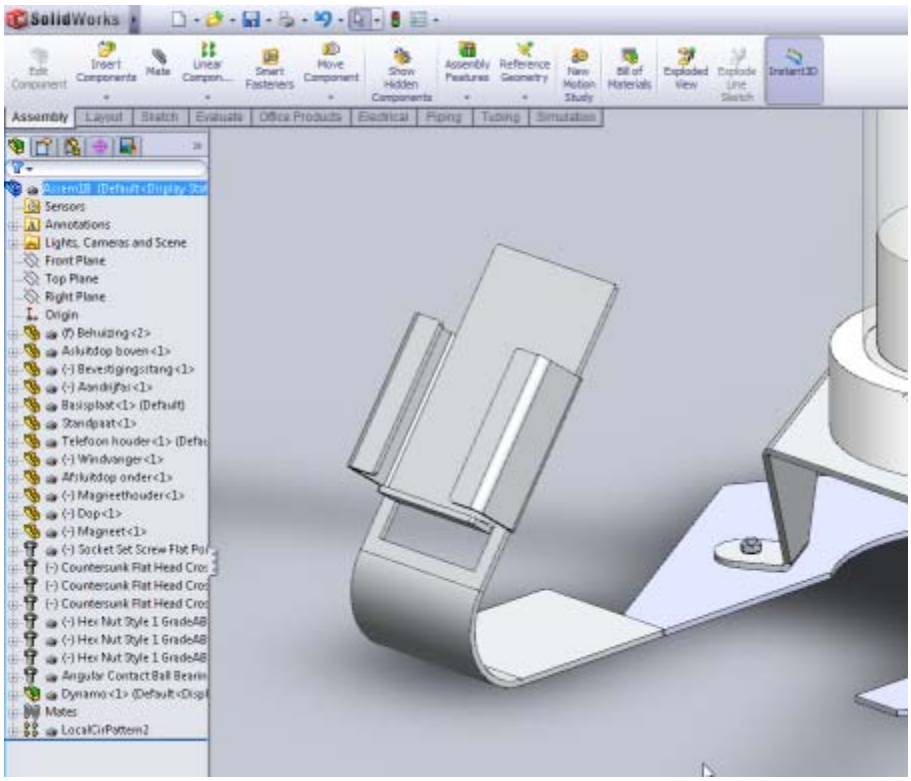
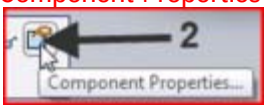
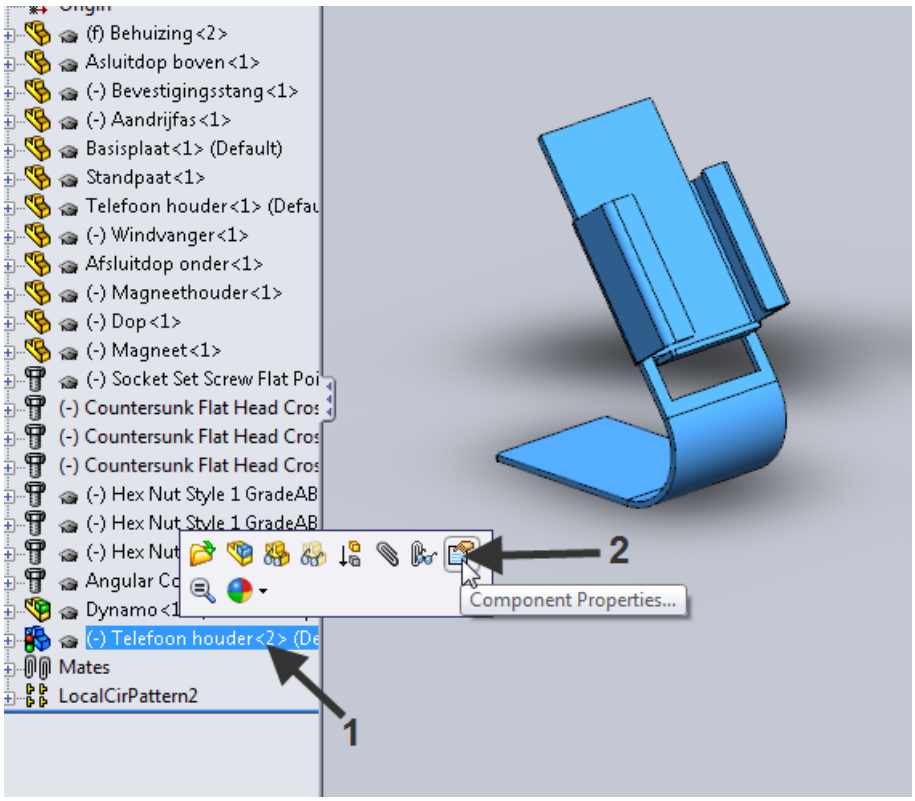
5. Drag 

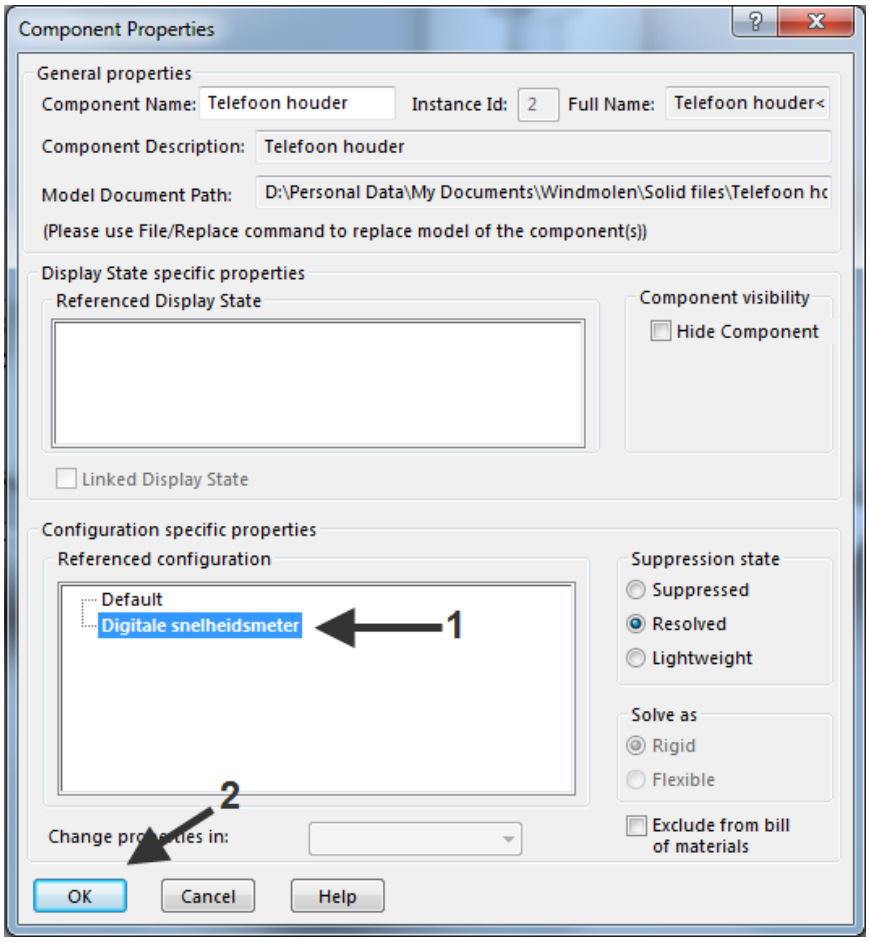
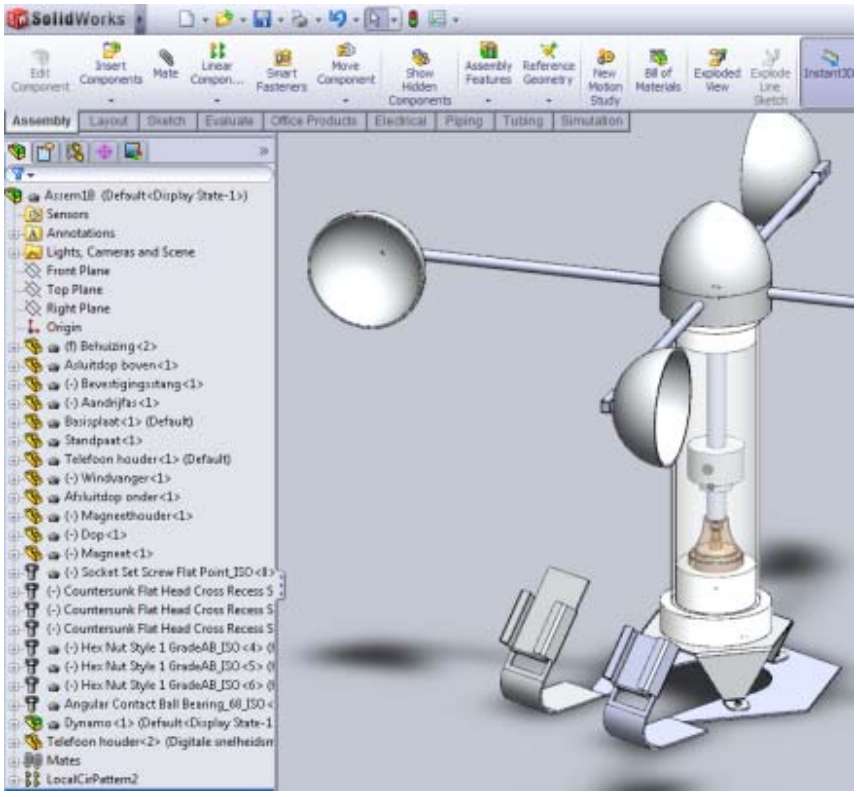
to the hole of the **top end.**

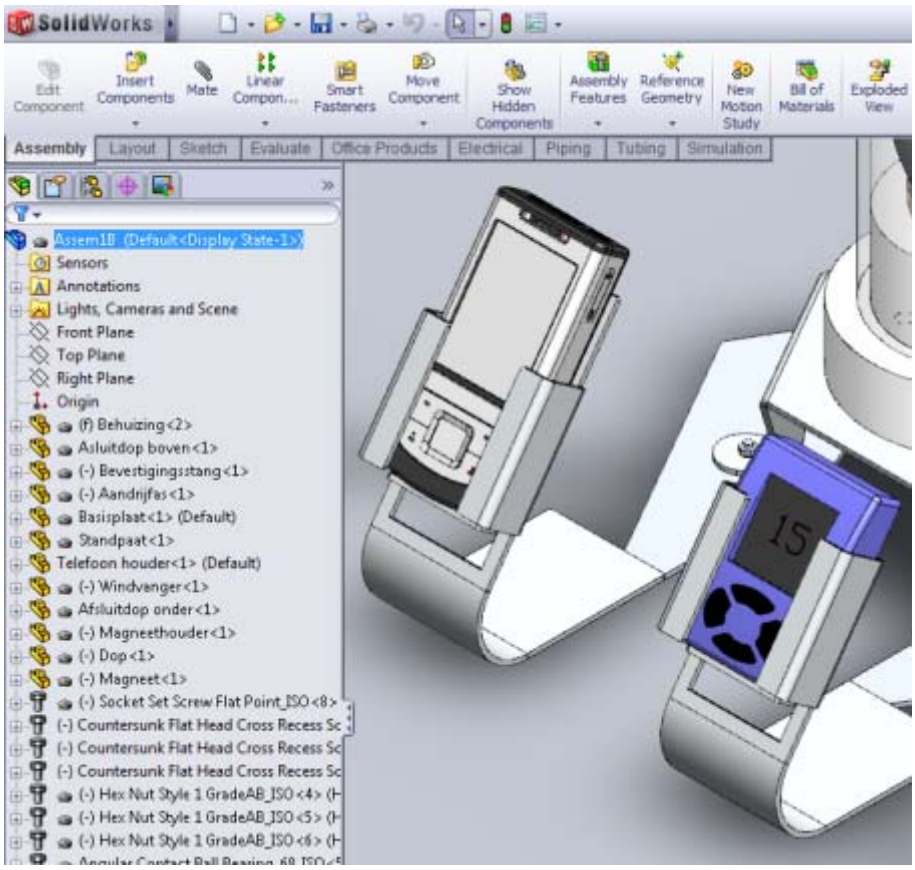
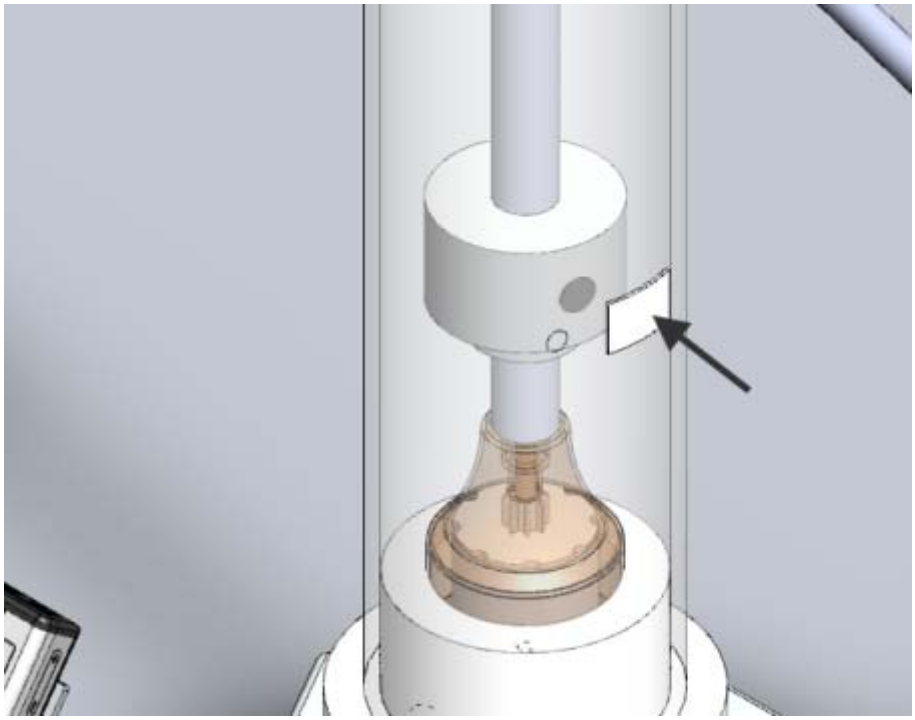


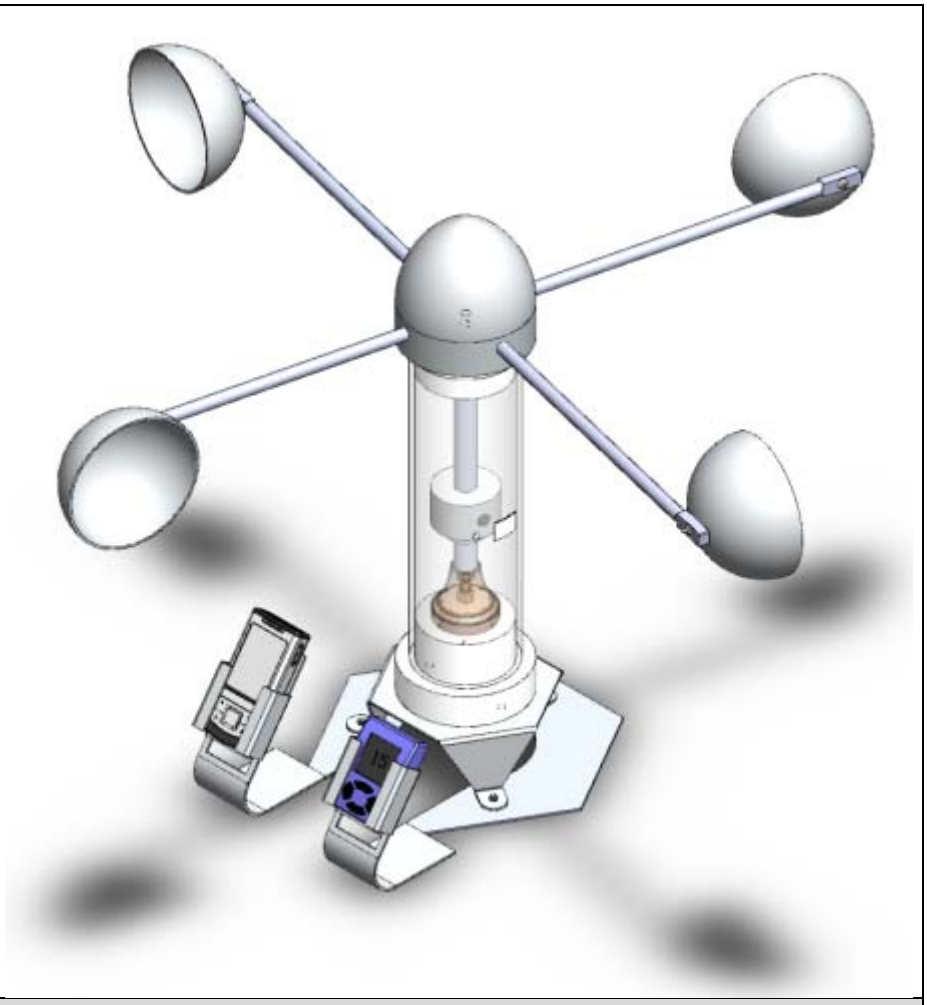
67	Choose the following: **1.** Rivet size:  **2.** Rivet length:  **3.** Click OK. In the next screen, click Cancel.	
68	Mate the rivet and the wing arm.	

69	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. 	
70	Choose the function: 	
71	**1.** Choose the axis of the cap  **2.** The window displays which axis you have selected.  **3.** Enter 360° degrees  **4.** In Property Manager, change the number of copies to 4  and check:  **5.** Select the windblade, the wing arm and the rivet.  **6.** Click OK.	

72	Mate the phone holder as in the figure.	
73	Add the phone holder piece once more. **1.** In: Feature Manager, click the  **2.** Click: Component Properties 	

74	A new menu is displayed. **1.** Choose: Digitale snelheidsmeter **2.** Then choose: OK	
75	Mate it in the same way as the phone holder. See the figure.	

76	Insert the phone and the speed indicator. Next, connect the phone and the speed indicator to the phone holders.	
77	Finally, add the magnet detector to the assembly. Then, mate the magnet detector, the housing and the bottom end.	

78	When the assembly is ready, save the file as **Windmill.sldasm**	
79		List the most important things you have learned during this tutorial.