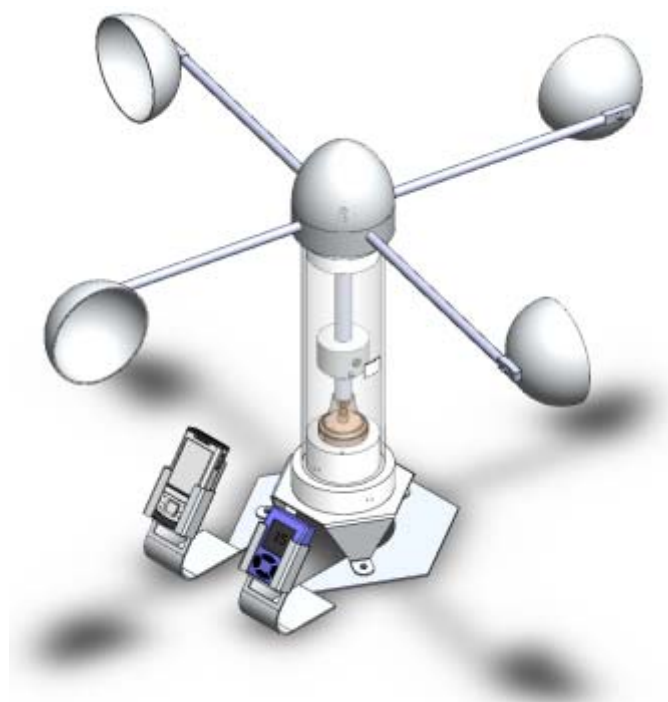


SolidWorks® tutorial 13-5 “Exciting”

Includes SolidWorks **SustainabilityXpress**



“A better world for our children and grandchildren”



Pre-vocational Secondary Education
and Senior Secondary Vocational Education



For SolidWorks® Educational Release 2009-2010

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U.S. Patents 5,815,154; 6,219,049; 6,219,055

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Initiative: Jack van den Broek and Nenad Raskovic

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

Completed by: Nenad Raskovic

A 3D perspective view of a dark gray, L-shaped bracket. The bracket features a semi-circular cutout in the center. It has three circular mounting holes: one on the top horizontal flange, one on the vertical flange, and one on the bottom horizontal flange. The bracket is shown against a white background with a soft shadow.



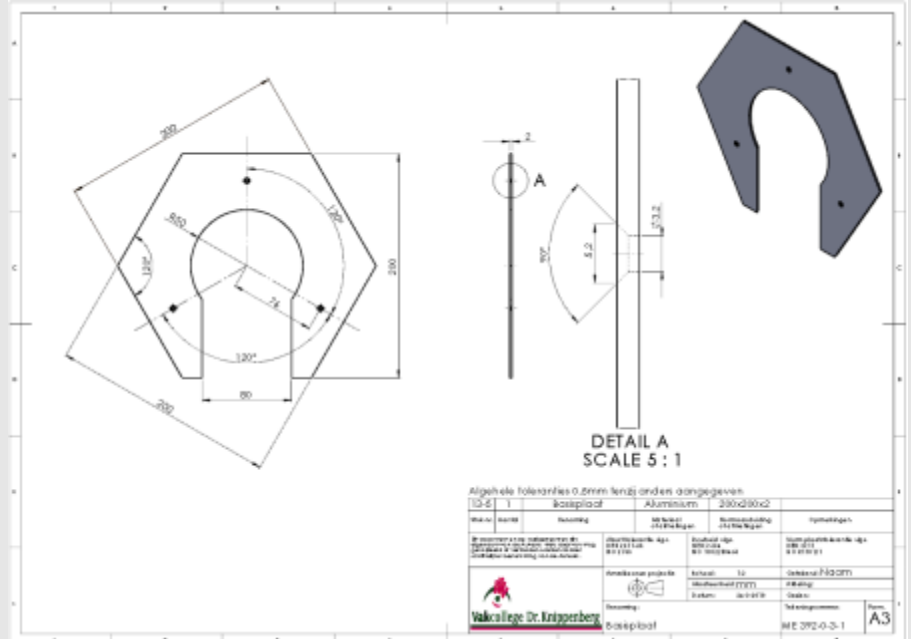
Base Sheet

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.

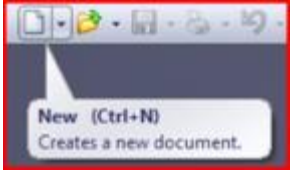
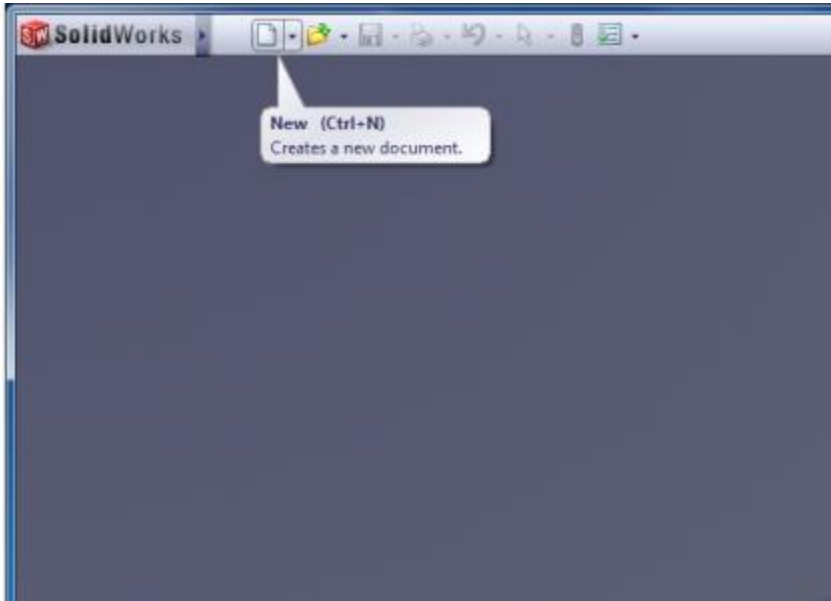


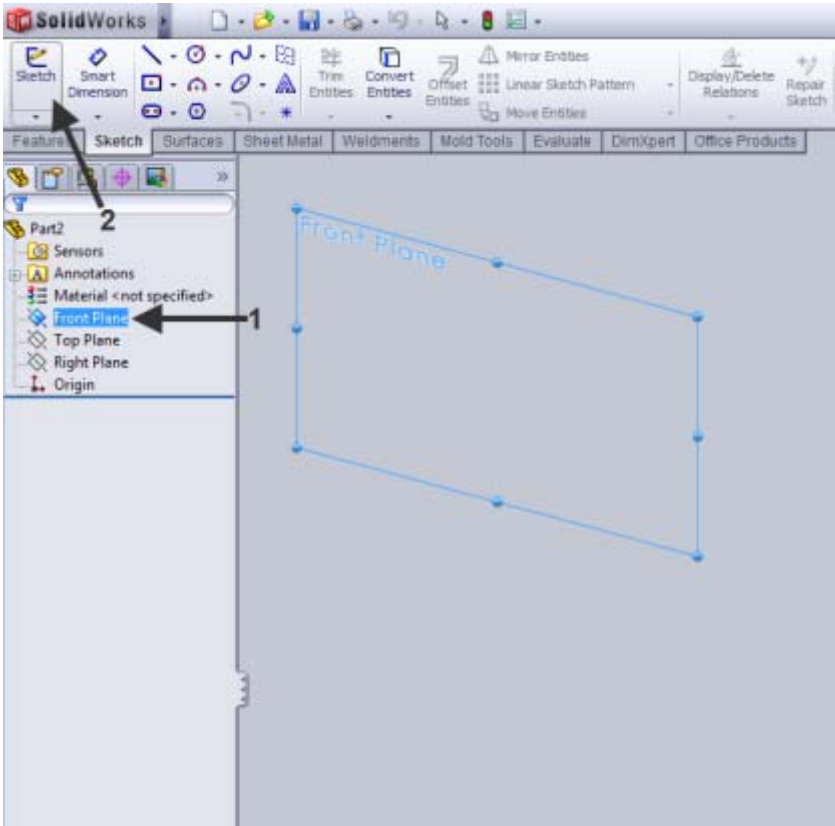


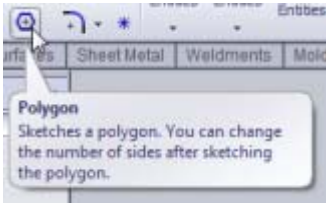
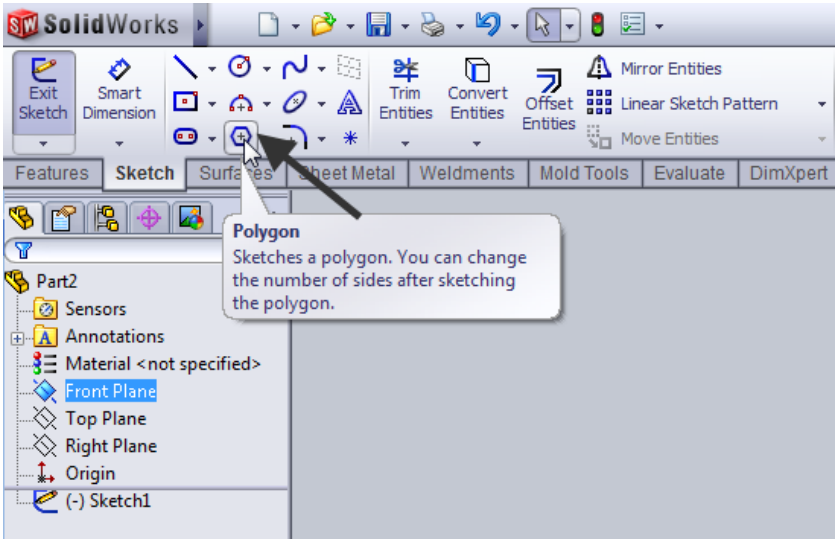

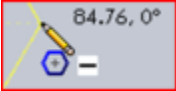

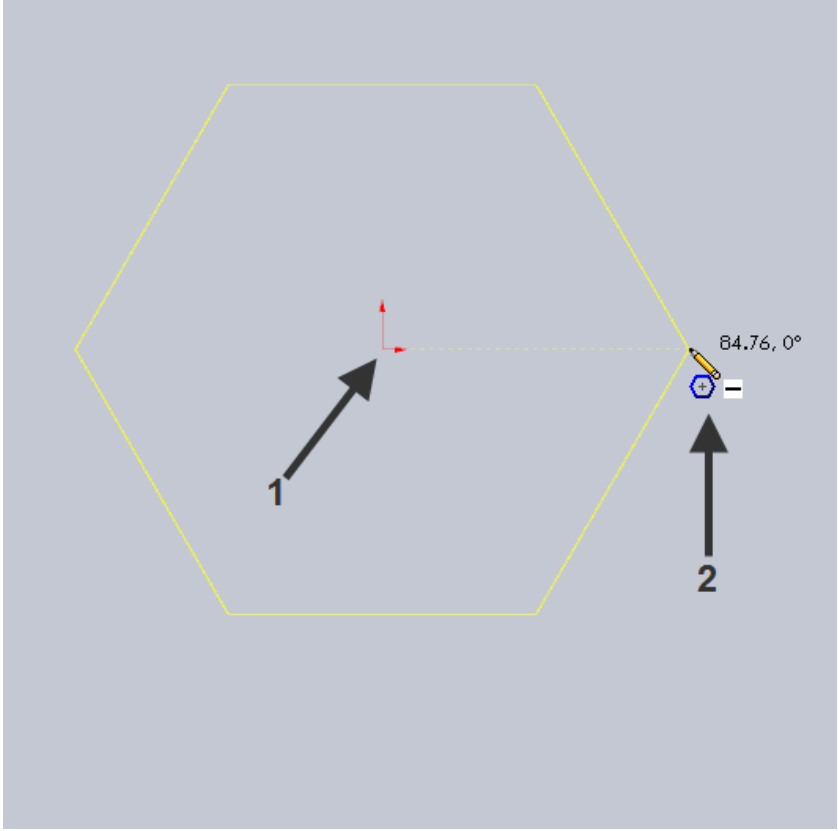
Work plan:

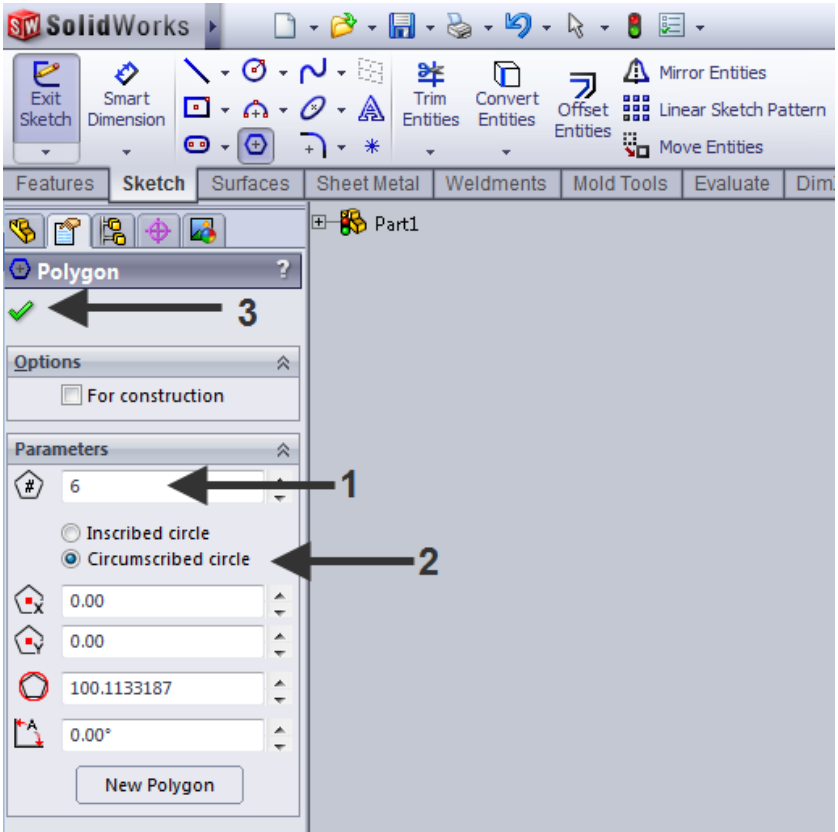
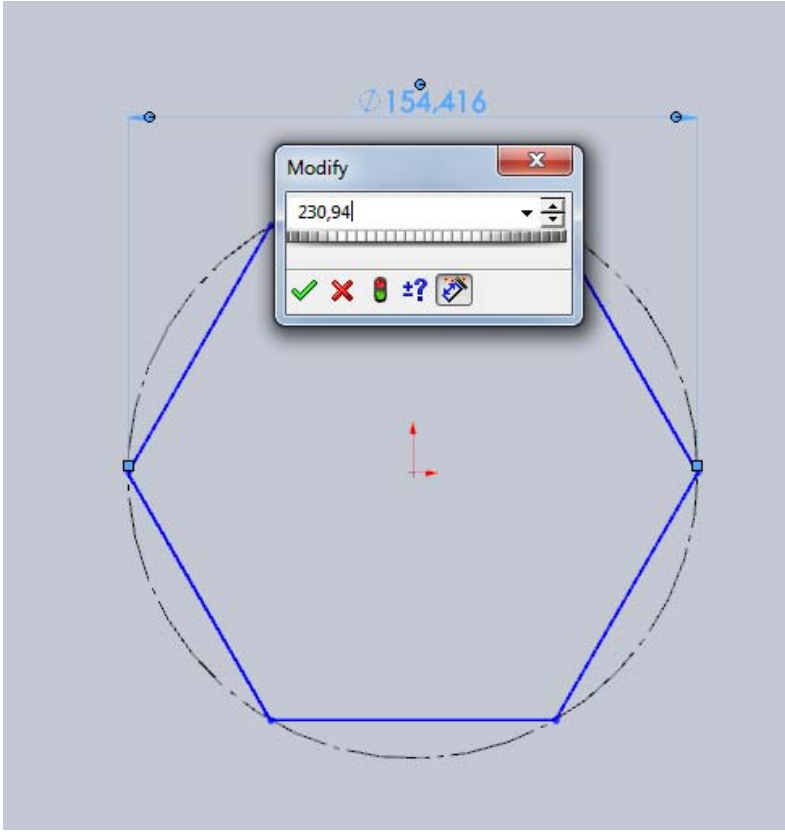
You will make the base sheet according to the drawing below.



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.

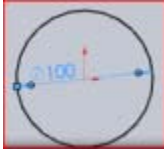
<p>1</p>	<p>Launch SolidWorks and open:</p>  <p>You do this to create a new document.</p>	
<p>2</p>	<p>1. Select:</p>  <p>2. Next, click on:</p>  <p>You do this to activate the Sketch environment.</p>	

<p>3</p>	<p>The base sheet is a regular hexagon.</p> <p>Click in the: Command-Manager on Polygon.</p>  <p>In this exercise, we will draw a regular hexagon.</p>	
<p>4</p>	<p>Draw a hexagon from point zero.</p> <ol style="list-style-type: none"> For the first point of the hexagon, click the  origin. For the second point, click at an arbitrary distance to the right of the origin.  Be sure to work horizontally! See icon:  	

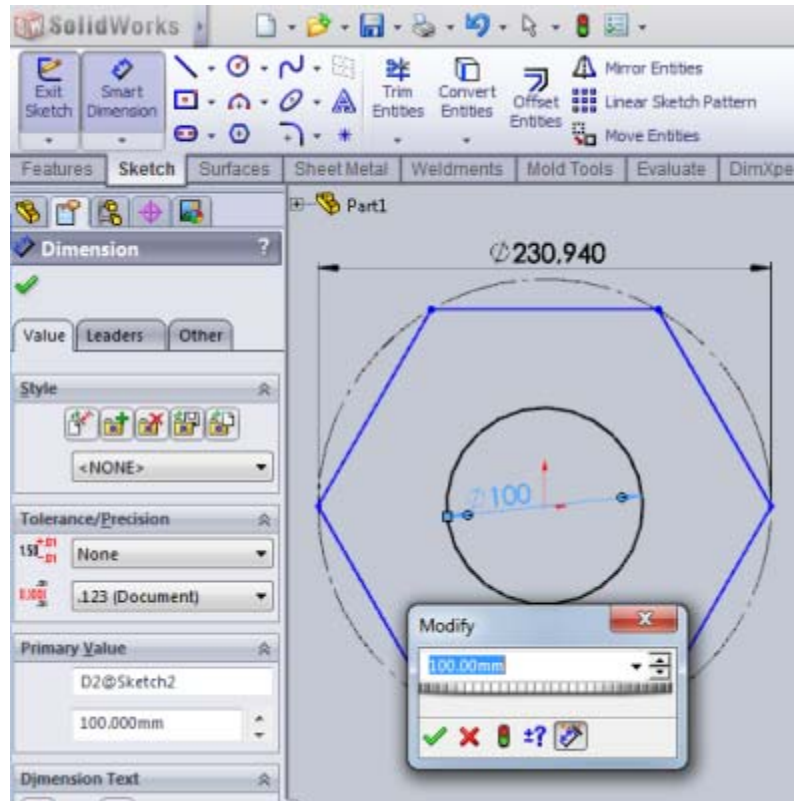
<p>5</p>	<p>Make sure that in the: Property Manager, under:</p> <p>Parameters</p> <ol style="list-style-type: none"> 1. The number of sides is set to 6 2. A circumscribed circle now determines the size. 3. Click OK. 	
<p>6</p>	<p>The size of the circle should become Ø 230.94mm.</p> <p>Because of the dimensioning of Ø 230.94mm, the size on the sides of the hexagon will be 200 mm.</p> <p>If you select the option:</p> <p>Inscribed circle</p> <p>Then the size displayed for the circle should be 200mm.</p> <p>Dimension it with the Smart Dimension function.</p>	

7

Draw a $\varnothing 100\text{mm}$ circle from the origin:



Next, dimension the circle.



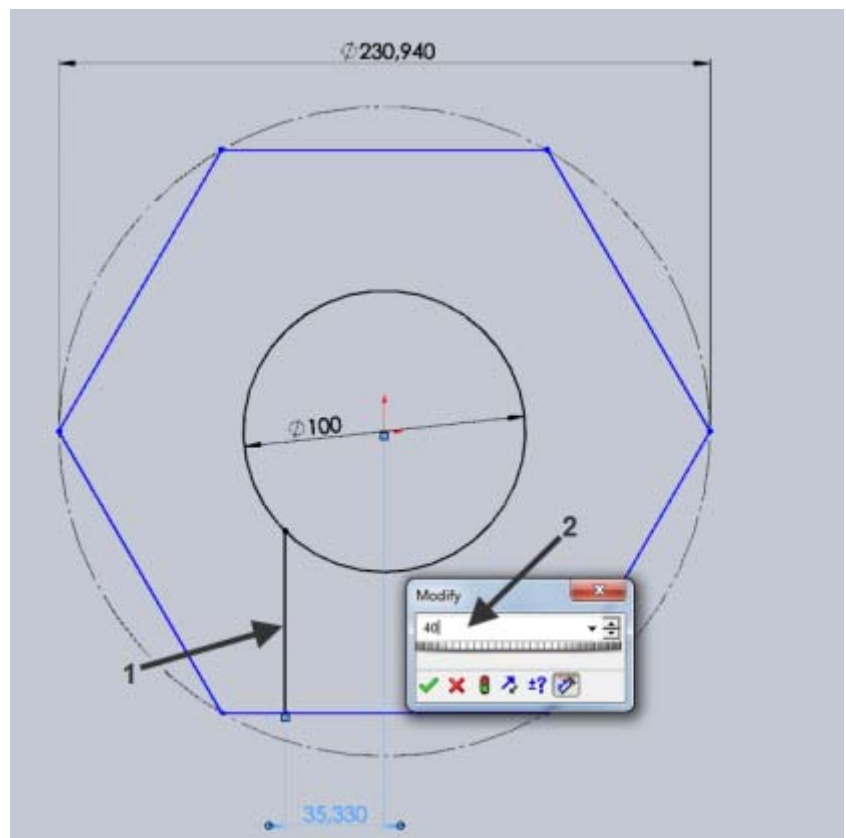
8

1. Now, draw a **Line**



straight to the top, as in the figure

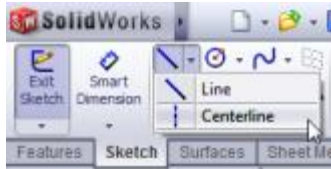
2. Then, dimension this line as in the adjoining figure.
The distance between the line and the center of the hexagon is **40 mm**.



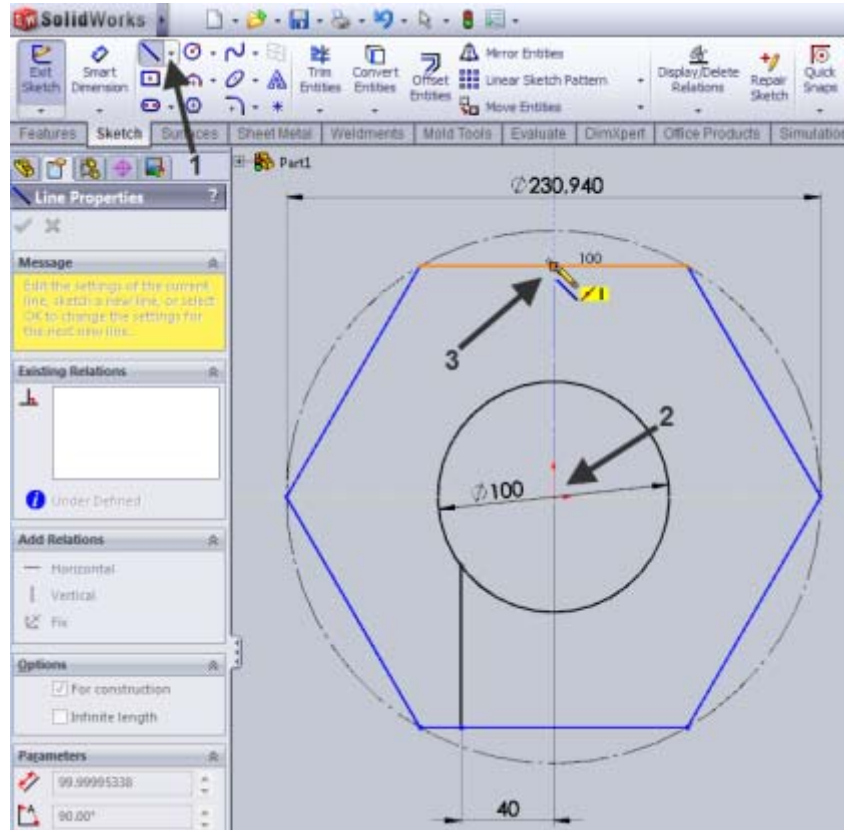
9

Draw a **Centerline** from the **origin** straight to the top.

1. Click the function: **Centerline**:



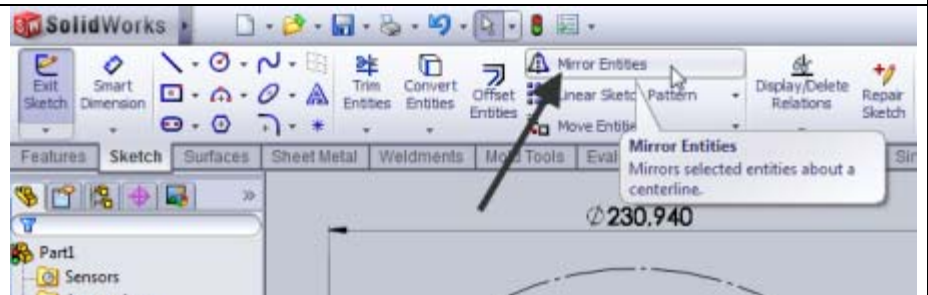
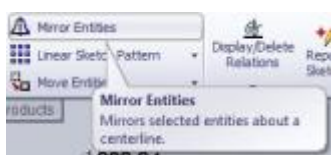
2. For the first point, click the **origin**.
3. Then, draw a Centerline straight to the top, as in the figure.

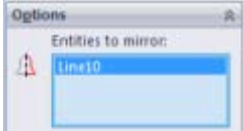
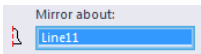

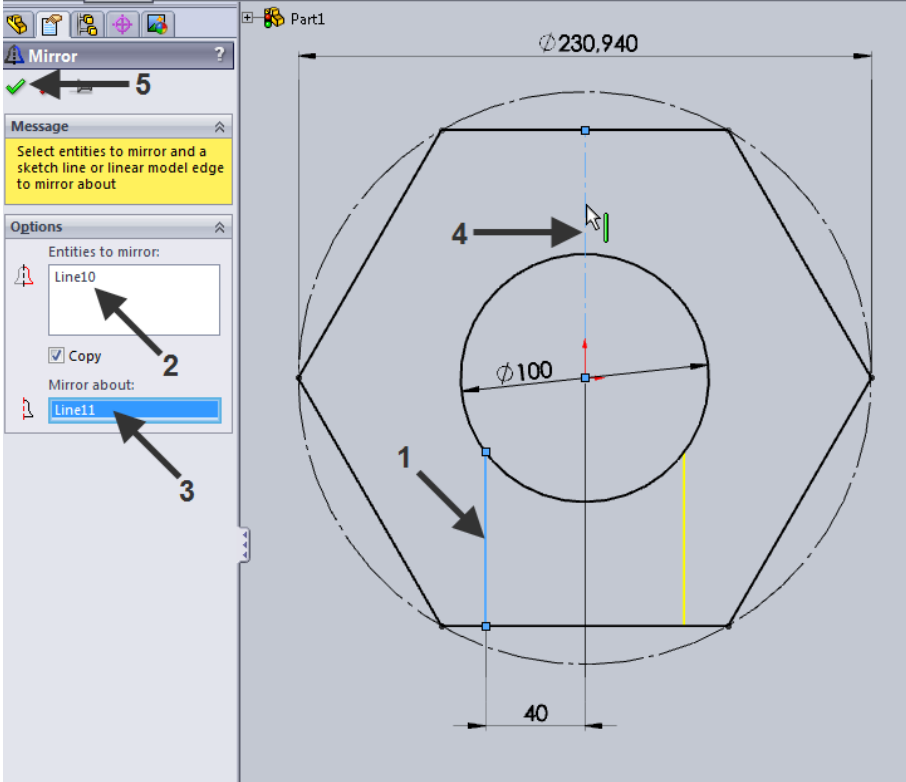
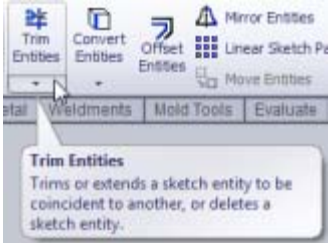
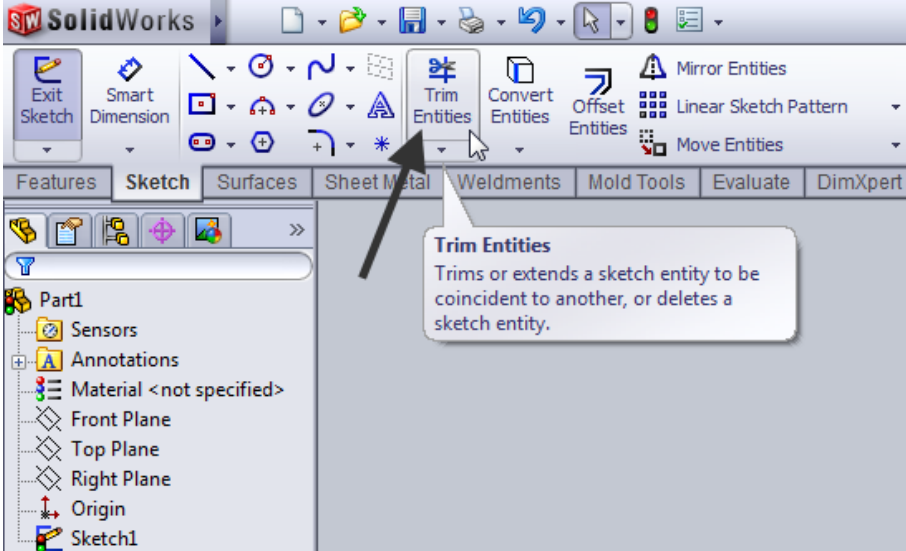




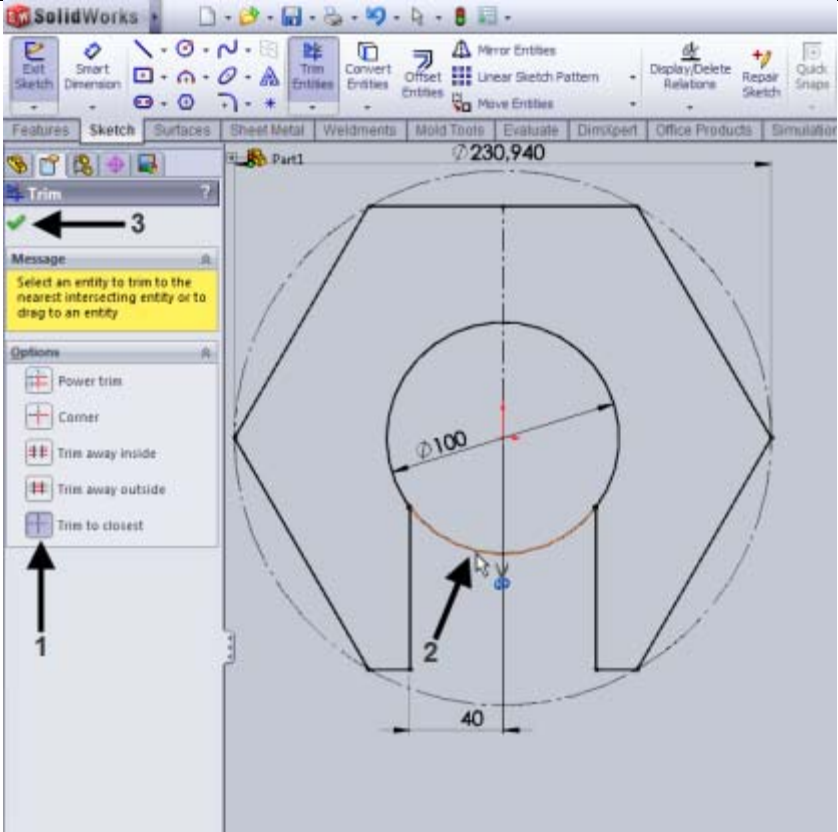
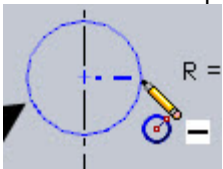
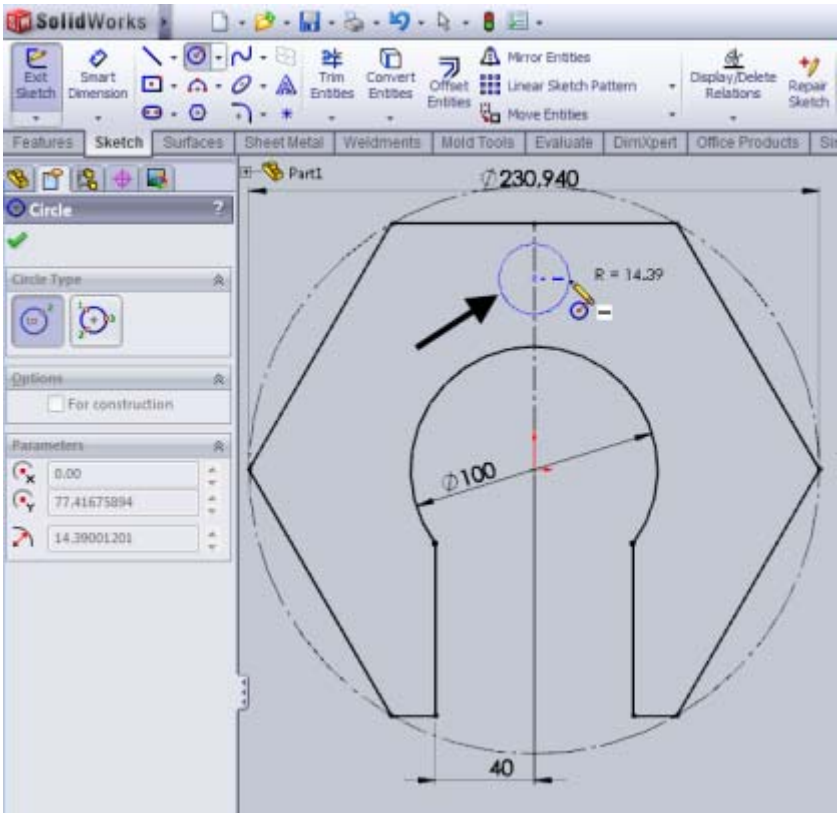
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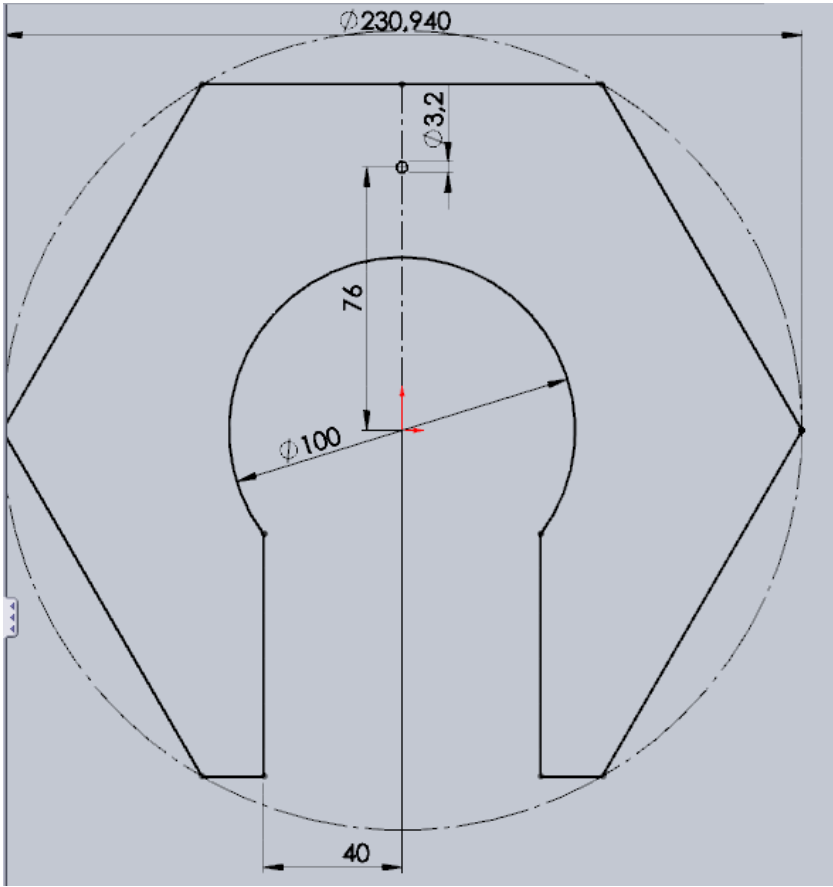

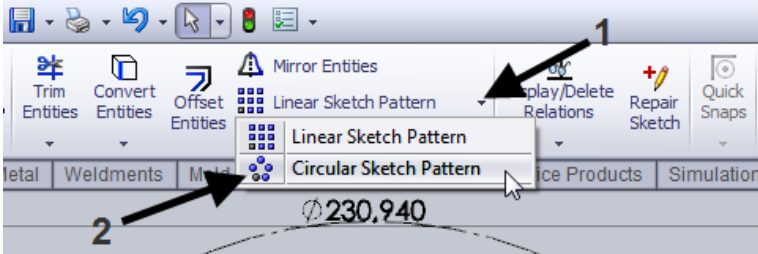
We will now mirror the line.


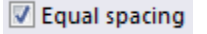

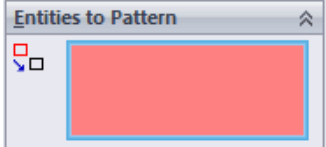
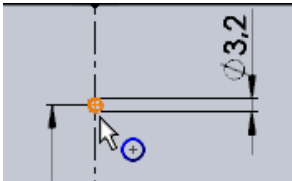
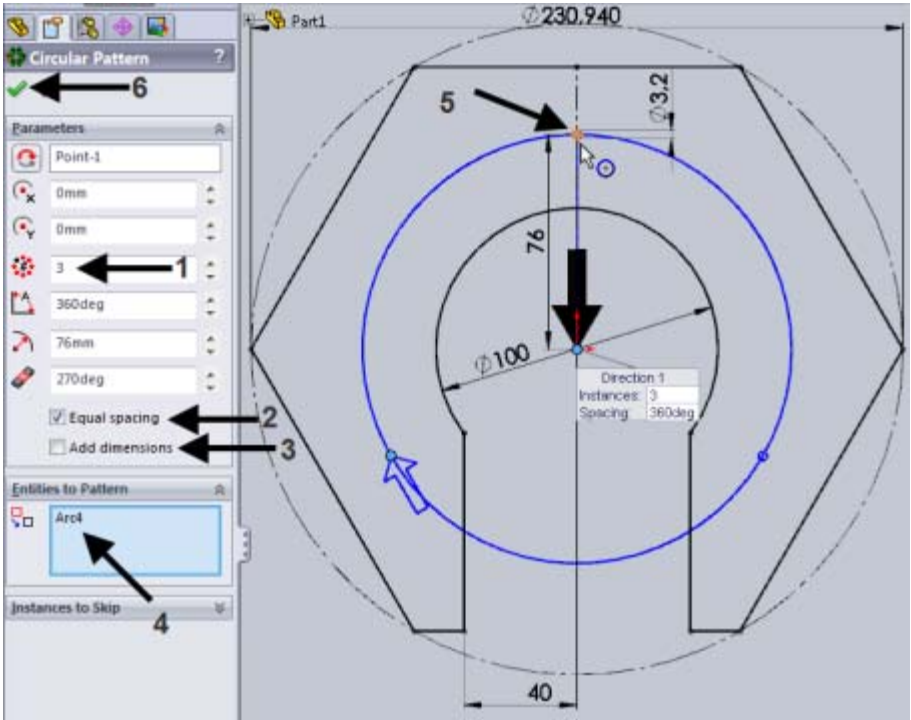

Select: **Mirror Entities**



<p>11</p>	<p>The Property Manager menu under Options shows the selected components you want to mirror.</p> <ol style="list-style-type: none"> 1. First, select the line. 2. The window Entities To mirror displays the selected component, in this case, the line.  <ol style="list-style-type: none"> 3. Click in the window: Mirror about: 4. Select the Centerline. Once you have selected the Centerline, it is displayed in the window Mirror about.  <ol style="list-style-type: none"> 5. Click OK.  	
<p>12</p>	<p>We want to remove the bottom of the circle and the hexagon to make a recess there.</p> <p>The bottom of the circle and the hexagon can be removed with the Trim Entities function.</p>  <p>In the Command Manager, click Trim Entities</p>	

<p>13</p>	<ol style="list-style-type: none"> 1. In the: Property Manager, select the option Trim to closest  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.  	
<p>14</p>	<p>Now, draw a circle. The center of the circle must be on the centerline. Look at the example.</p> 	

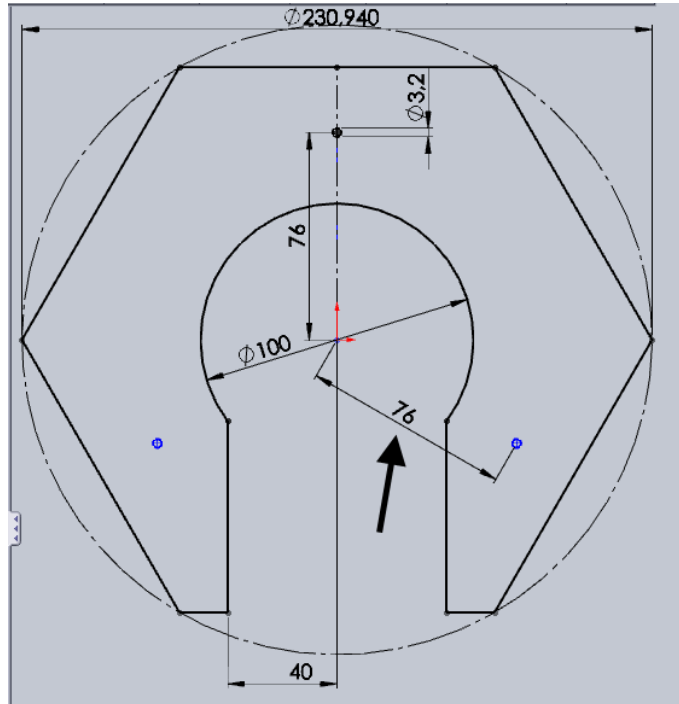
<p>15</p>	<p>Enter the circle dimensions. The circle has a diameter of 3,2mm and a height of 76mm as measured from the origin.</p>	 <p>Technical drawing of a mechanical part. The drawing shows a cross-section of a part with a central circular feature. The outer diameter is labeled as $\varnothing 230,940$. The inner diameter of the central feature is labeled as $\varnothing 100$. The height of the central feature is labeled as 76. A small circle, likely representing a hole or a fillet, has a diameter of $\varnothing 3,2$. The drawing is centered on a vertical axis, with a horizontal dimension of 40 indicated at the bottom.</p>
<p>16</p>	<p>We want to add two more circles to the Sketch. To do this, use the command Circular Sketch Pattern.</p> <ol style="list-style-type: none"> 1. First, click the arrow to expand the menu. 2. Select <p> Circular Sketch Pattern</p>	 <p>Screenshot of the SolidWorks software interface. The 'Sketch' menu is open, and the 'Circular Sketch Pattern' command is highlighted. An arrow labeled '1' points to the 'Sketch' menu, and an arrow labeled '2' points to the 'Circular Sketch Pattern' command. The background shows a technical drawing with a dimension of $\varnothing 230,940$.</p>

<p>17</p>	<p>Now, do the following:</p> <ol style="list-style-type: none"> 1. Set the number of copies to 3.  <ol style="list-style-type: none"> 2. Check Equal Spacing.  <ol style="list-style-type: none"> 3. Uncheck Add dimensions.  <ol style="list-style-type: none"> 4. Click anywhere in the window Entities to Pattern.  <p>The window is empty at first, but as soon as you click the components to be copied, the window displays the selected components.</p> <ol style="list-style-type: none"> 5. Select a 3.2mm circle.  <ol style="list-style-type: none"> 6. Click OK. 	
		<p>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.</p>

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.



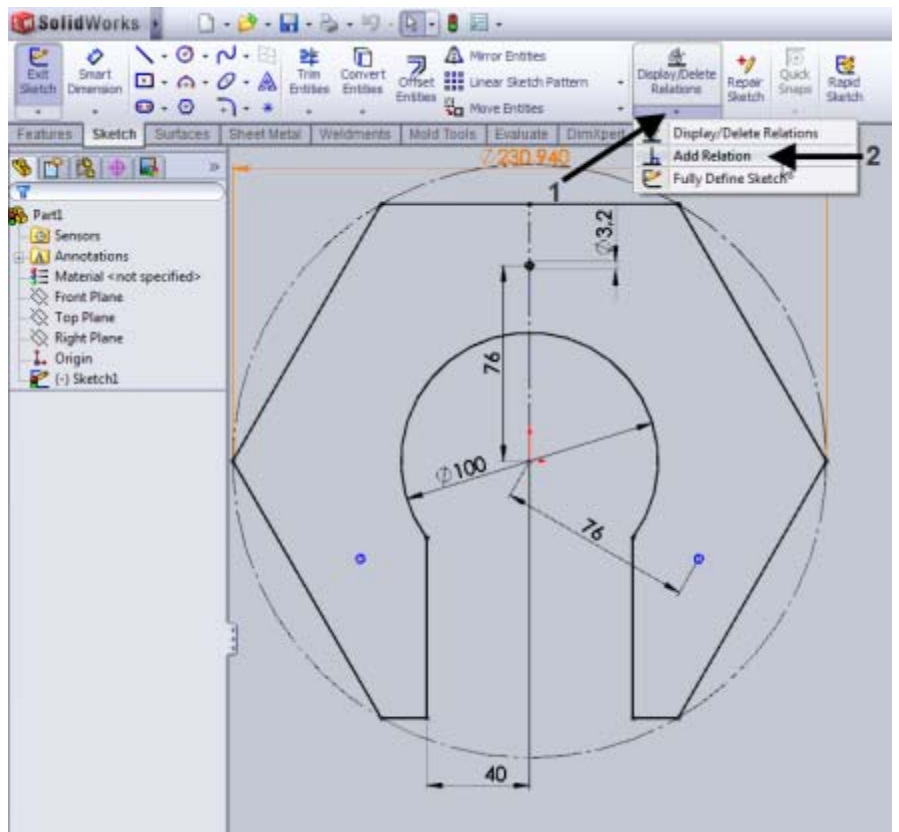
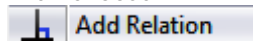
19


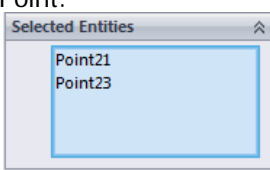
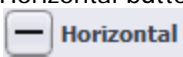
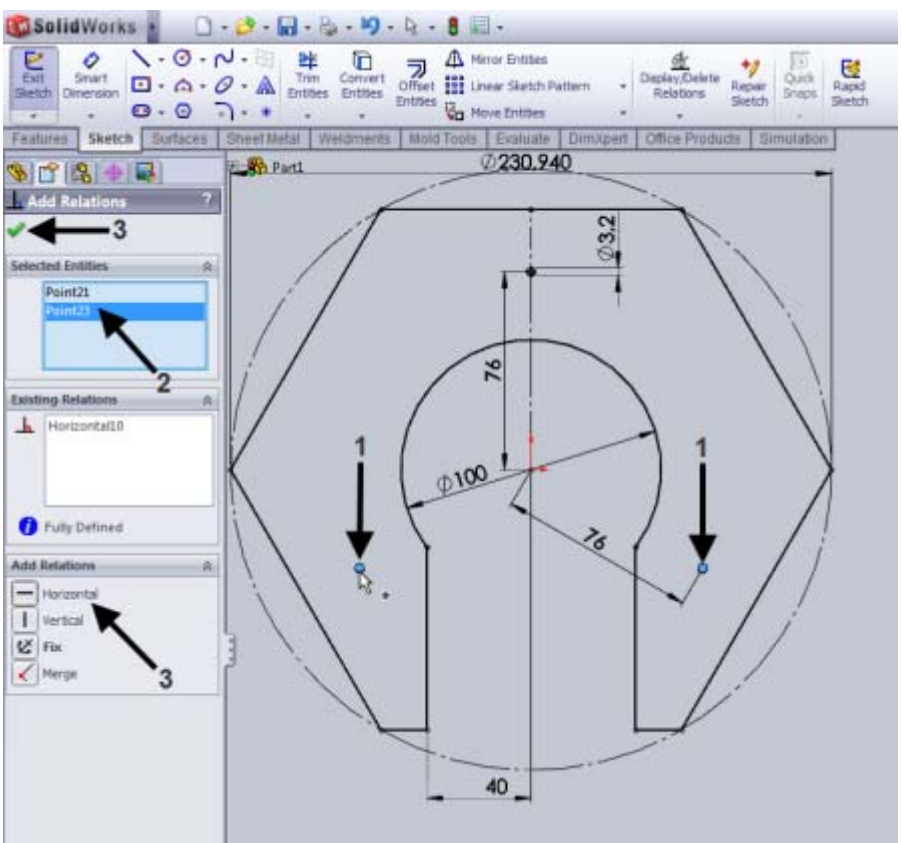



The sketch still hasn't been completely defined yet. Adding dimensioning and/or relations provide a completely defined sketch. In this case, you want to add a relation.

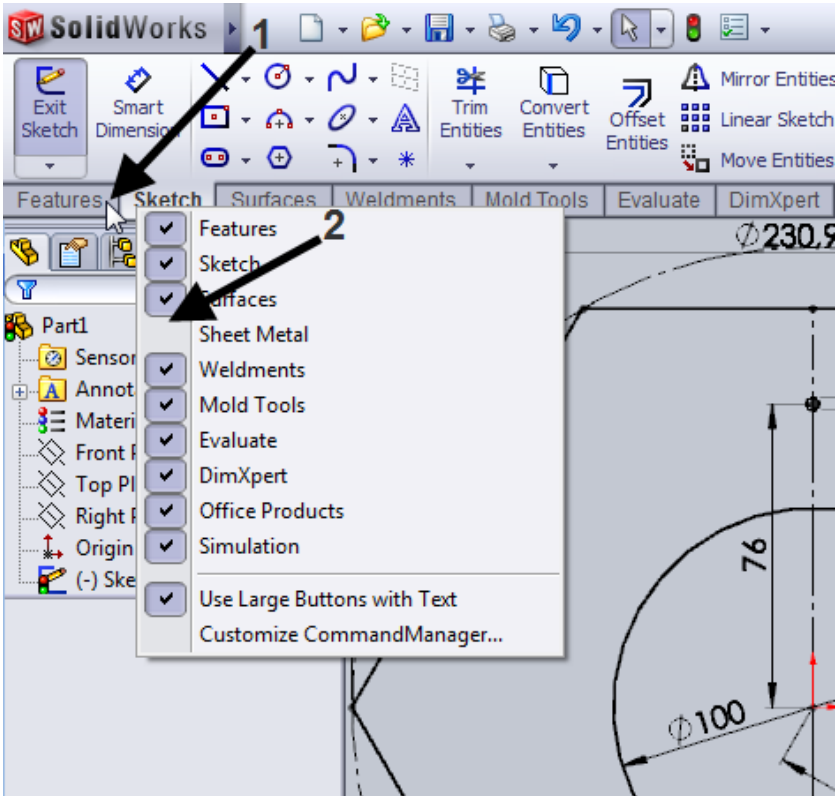

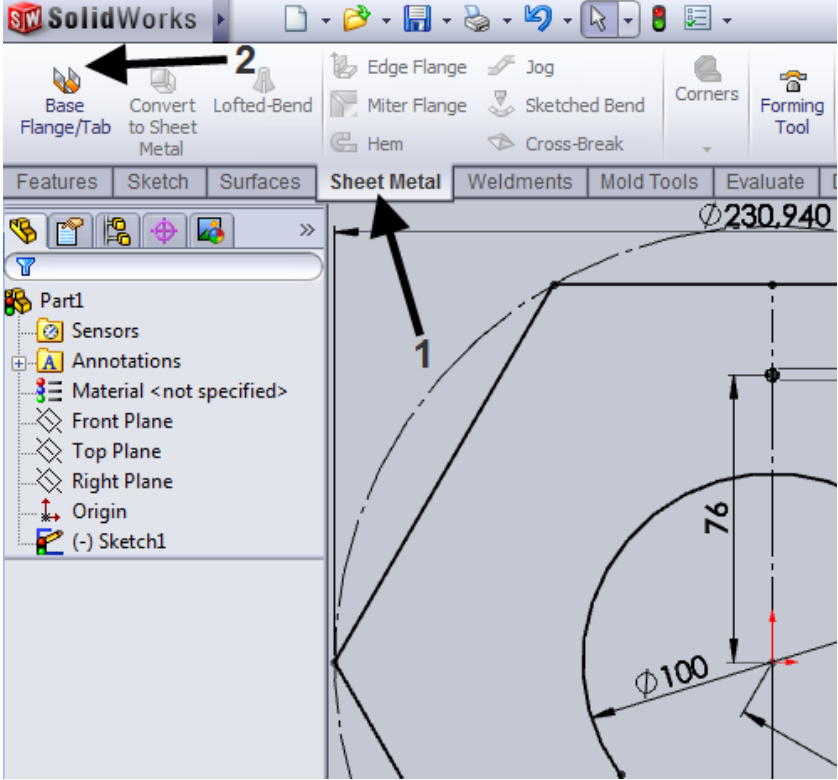
1. Click the arrow **Display/Delete Relations**



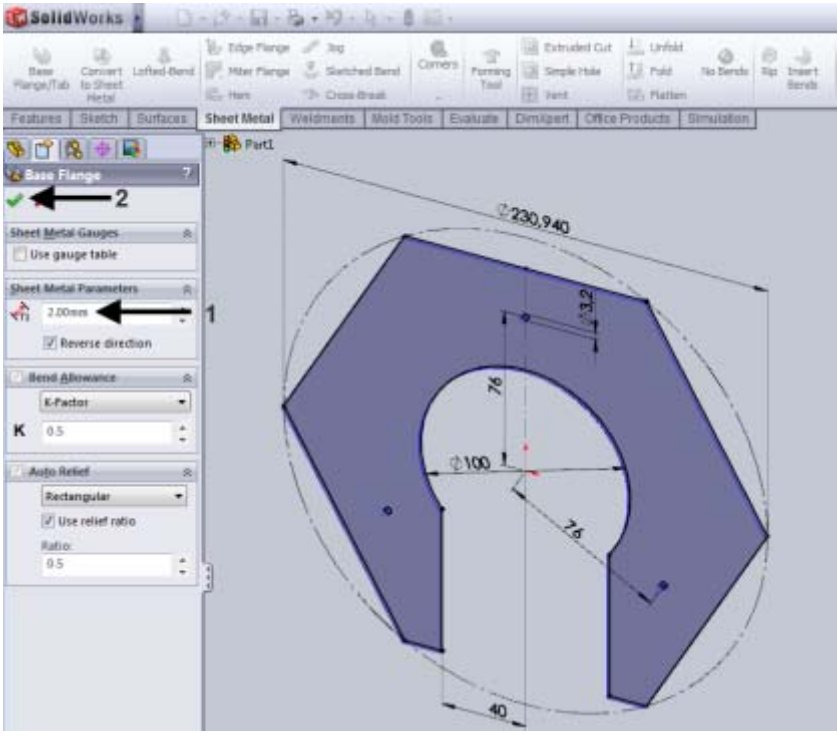


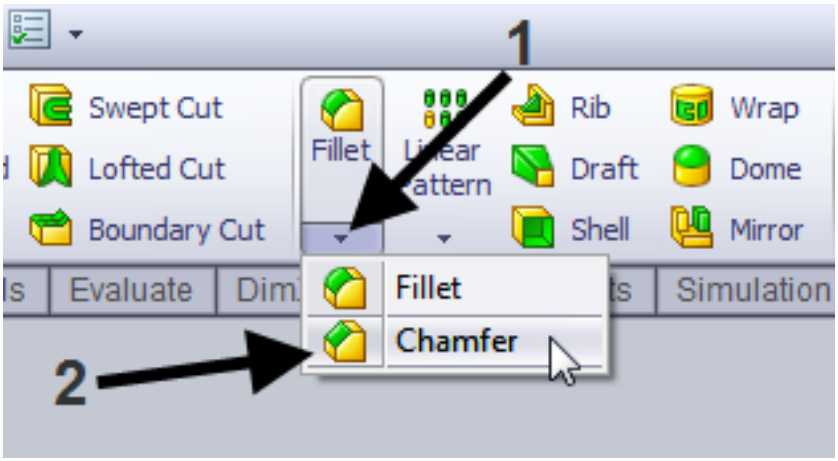



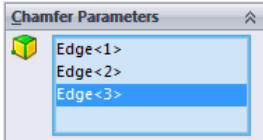
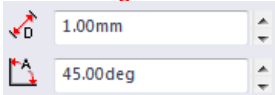
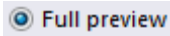

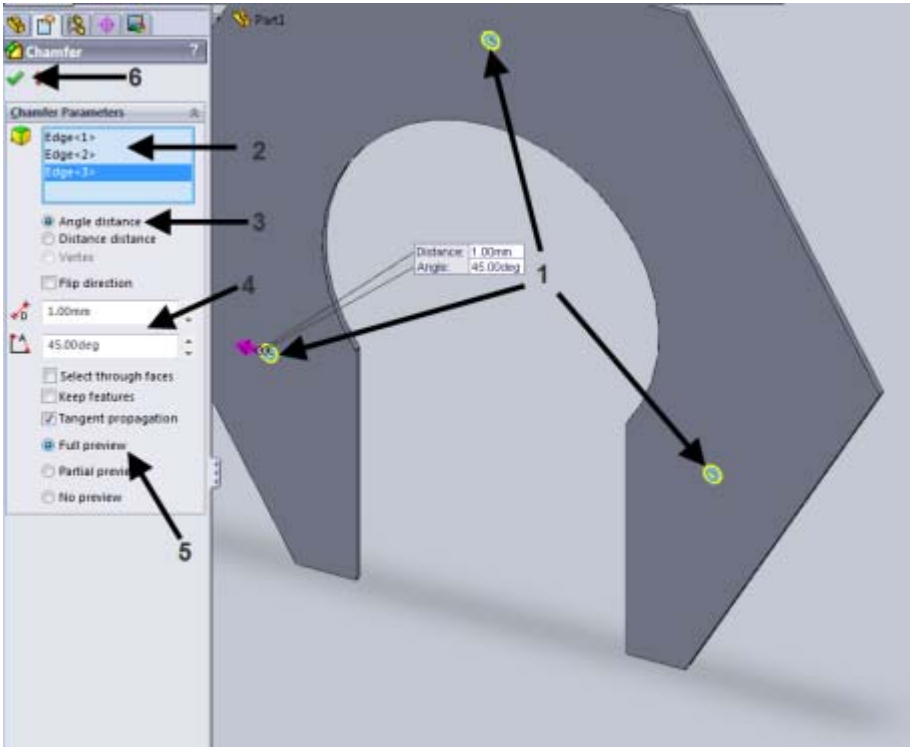
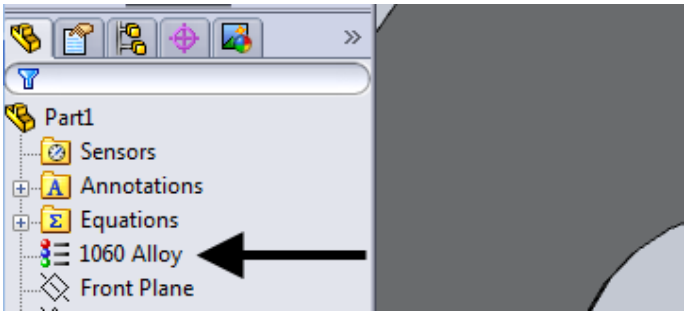
2. Then choose



<p>20</p>	<ol style="list-style-type: none"> 1. Select the center of the two lower circles.  <p>The plus sign shows you will be selecting the center.</p> 2. The selected objects will appear in the blue window, in this case, Point.  3. Then, click the Horizontal button  <p>This will put both circles on one line.</p> 	
		<p>The function  Add Relation 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.</p> <p>To display and/or remove all existing relations, use the function  Display/Delete Relations.</p>

<p>21</p>	<p>First, make sure the Sheet Metal buttons are available.</p> <p>The best way to do this is to add them to the Command Manager.</p> <ol style="list-style-type: none"> 1. With the right mouse button, click a tab in Command Manager. 2. In the displayed menu, click: Sheet Metal. <p>ATTENTION!! If Sheet Metal is already checked in your system, continue to step 20.</p>	
<p>22</p>	<ol style="list-style-type: none"> 1. In: Command Manager, first click: Sheet Metal. 2. Then, click Base Flange/Tab. 	

<p>23</p>	<ol style="list-style-type: none"> 1. In: Property Manager, enter 2mm as material gauge.  2. Click OK.  <p>Leave the rest of the menu unchanged.</p>	
<p>24</p>	<p>Next, we will make the chamfer at the top of the circle.</p> <ol style="list-style-type: none"> 1. In the: Command Manager, click the arrow  under Fillet. 2. Click Chamfer.  	

<p>25</p>	<p>You must now set and verify a few things.</p> <ol style="list-style-type: none"> 1. Select the top edge of all three circles. (Use the CTRL key)  <ol style="list-style-type: none"> 2. In the blue area, Edge has now been selected three times.  <ol style="list-style-type: none"> 3. Make sure the 'Angle distance' option is selected. If not, check it. Angle distance <ol style="list-style-type: none"> 4. As the chamfer distance, enter 1mm and 45 deg.  <ol style="list-style-type: none"> 5. Make sure the option 'Full preview' is selected.  <p>The model shows exactly what will happen.</p> <ol style="list-style-type: none"> 6. If everything has been set correctly, click OK. 	
<p>26</p>	<p>Change the material to 1060 Alloy.</p> <p>The model is now ready.</p> <p>In the toolbar, click Save and name the file: Base Sheet</p>	

SolidWorks Sustainability Xpress

"A better world for our children and grandchildren"



As a developer/designer, you must take several aspects into account.


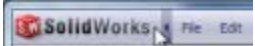






One of these aspects is the environmental impact of your design.

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.

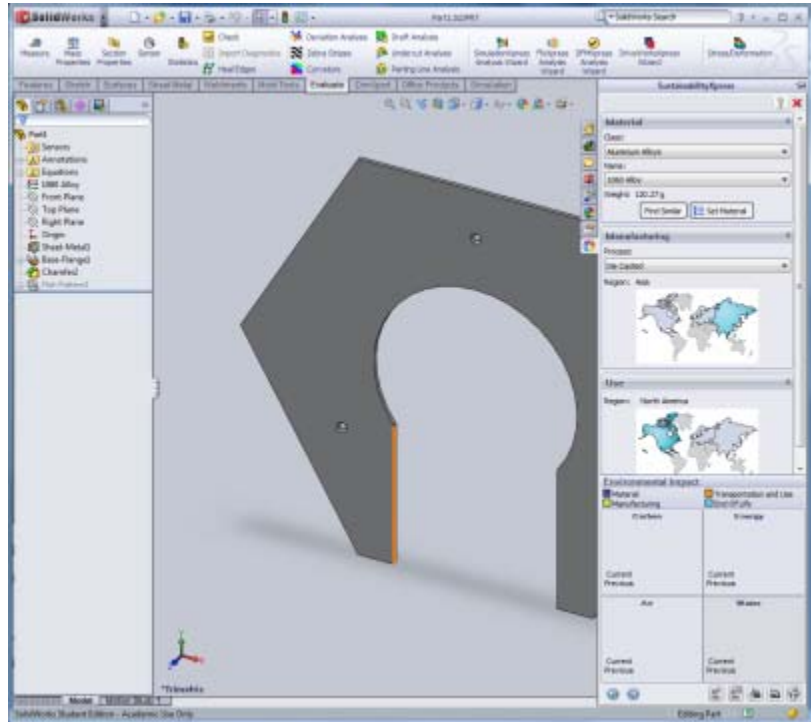
<p>27</p>	<p>Before continuing to  SustainabilityXpress..., please make sure this SolidWorks function has been installed on your computer.</p> <ol style="list-style-type: none"> 1. To open the menu, click the arrow in the Menu Bar:  2. Next, click Tools:  3. Make sure the function:  SustainabilityXpress... has been installed. 4. If the function: Sustainability Xpress has already been installed, continue to the next step. Otherwise, follow the instructions below. 	
	 	<p>If you are using SolidWorks student Edition version 2009-2010, you must download the application from the following address:</p> <p>http://www.solidworks.com/sw/education/7375_ENU_HTML.htm</p> <p>Be sure you download the appropriate version, as there are two versions for Windows: 32 bits and 64 bits.</p> <p>If you are using SolidWorks student Edition version 2010-2011, you will find SustainabilityXpress under the tab Evaluate (1) then select SustainabilityXpress (2).</p> 

28

After launching the function:



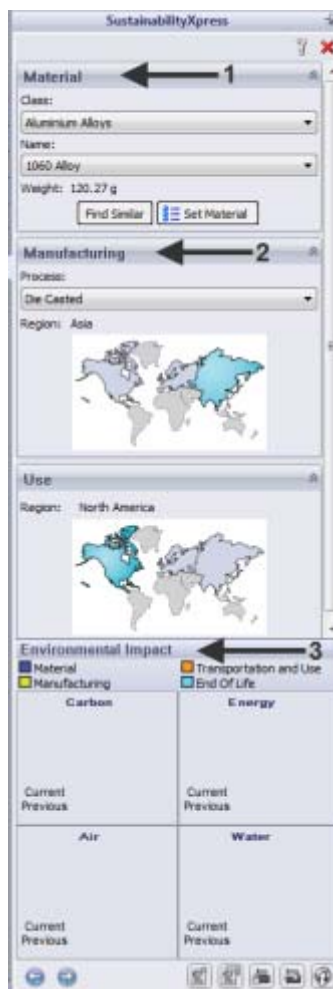
SolidWorks opens a new window to the right.



29

The window includes three important areas:

1. **Material** Enter the material properties here.
2. **Manufacturing** Specify 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.

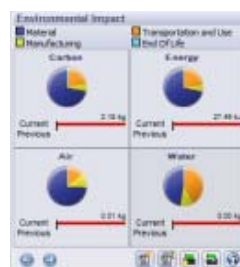
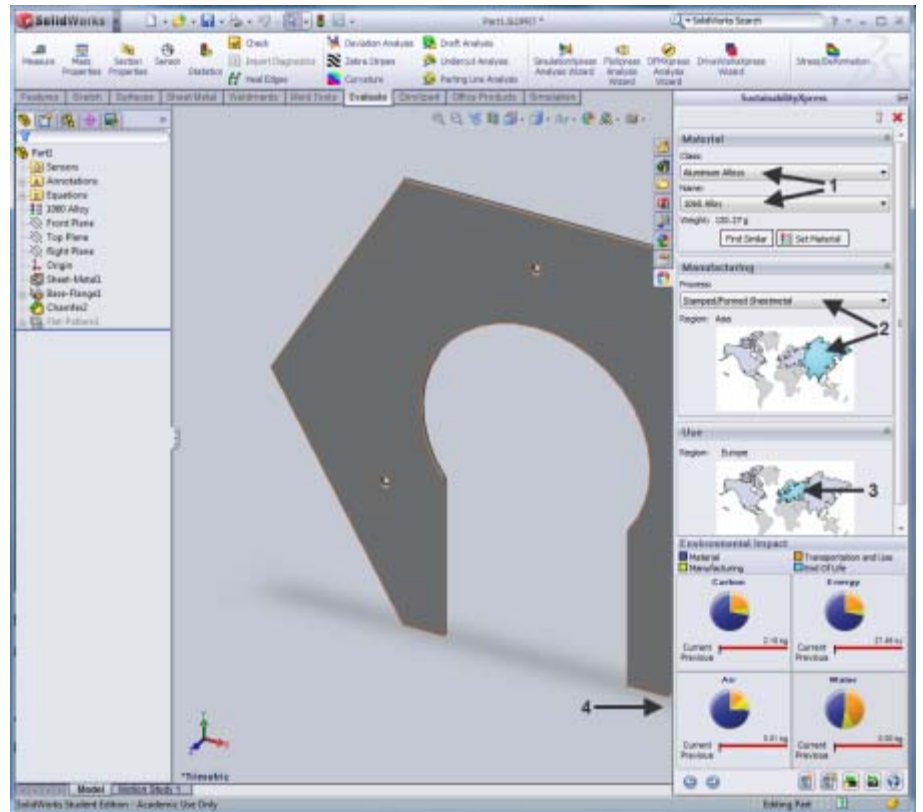


2. In the window: **Manufacturing** you enter this as: **Process:** **Stamped/Formatted Sheetmetal**

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.

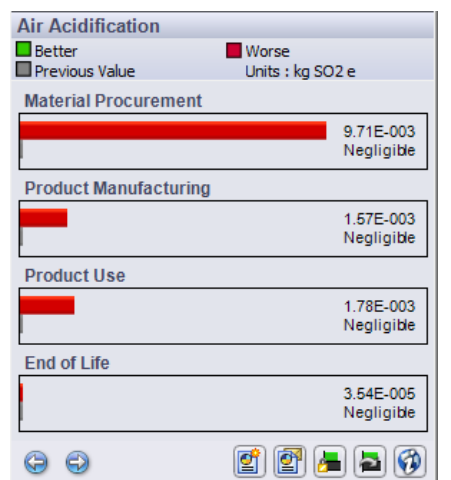
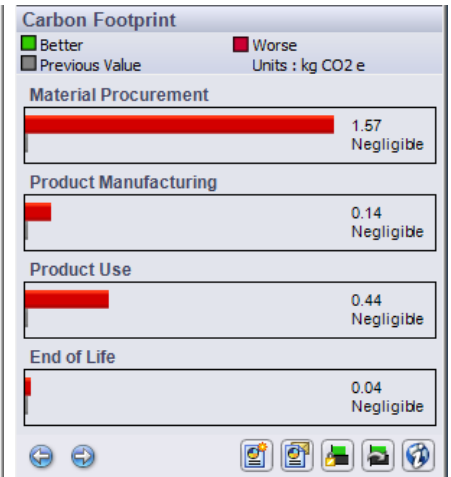
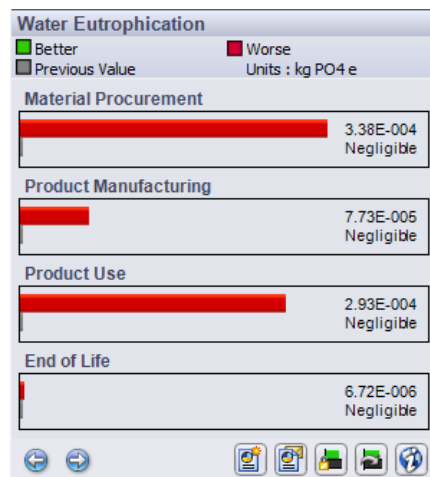
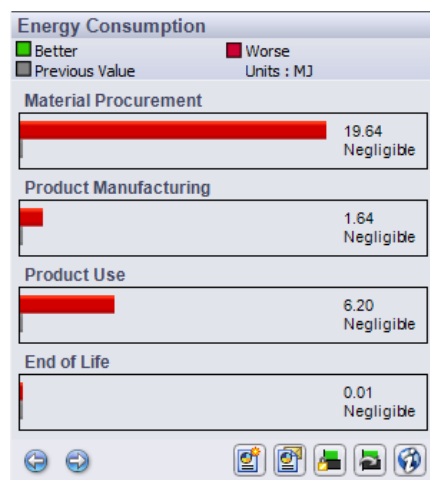
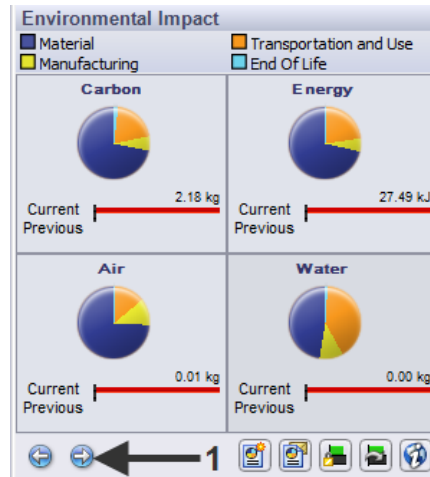
Air Acidification


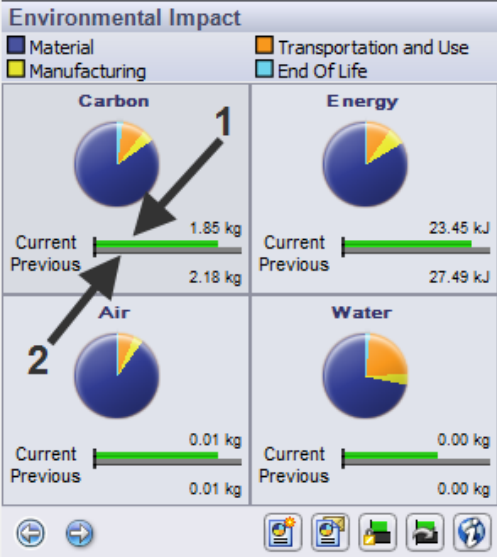
Impact on the air! In particular, contribution to acid rain.

E.g.

Water Eutrophication

Impact on water! Resulting in algal growth in coastal waters.



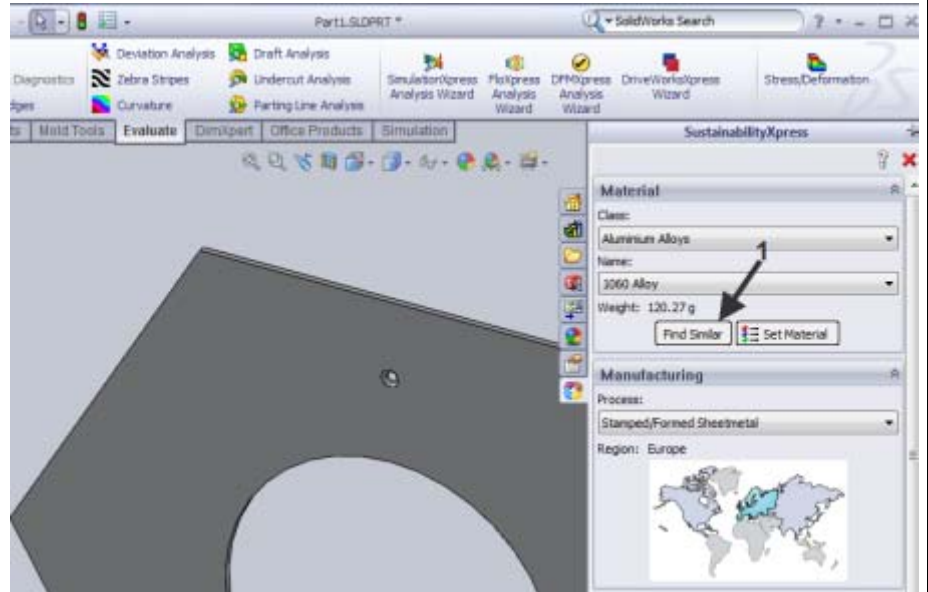
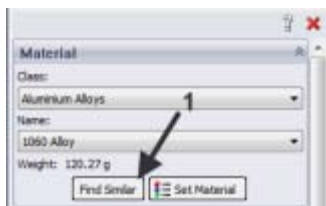
<p>32</p>	<p>Let us now change the production location to see how the environmental impact changes if the base sheet is not produced in Asia but rather somewhere else, for example in Europe.</p> <p>1. Change the: Region: into <u>Europe</u>.</p>	
<p>33</p>	<p>Now, watch the diagrams. There is a significant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

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.

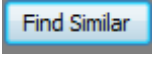
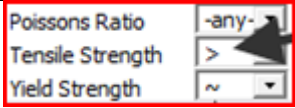

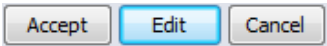
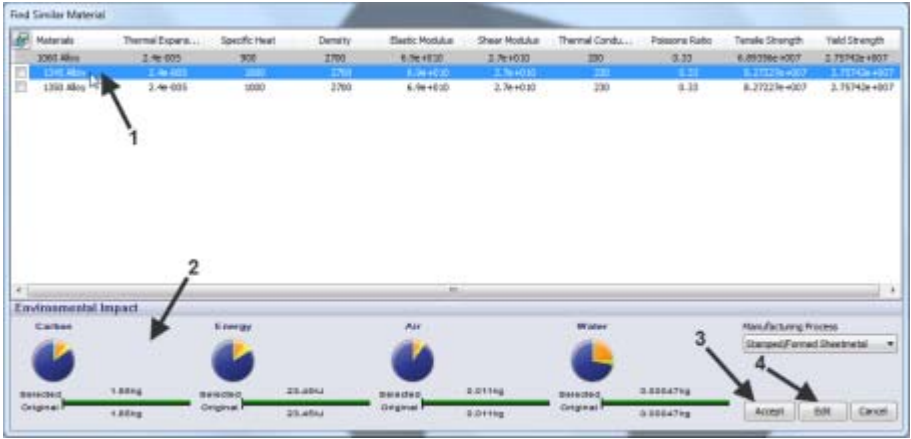
1. Enter Aluminum as the production material.

2. Scroll down to choose Aluminum.

Material Class = Aluminum

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



	<p>4. Let us choose a material with higher tensile strength. Click the scroll down menu and select greater than >.</p> <p>5. We will leave the yield value of the material unchanged ~.</p> <p>6. Click Find Similar</p> <p></p>	 
36	<p>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.</p> <p><input type="checkbox"/> 1345 Alloy</p> <p>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.</p> <p>3. You can now do the following: Accept, Edit or Cancel.</p> <p></p> <p>4. Let us choose Edit because we want to know what will happen if we choose steel instead of aluminum.</p>	

37

1. Choose **Steel** as the material.

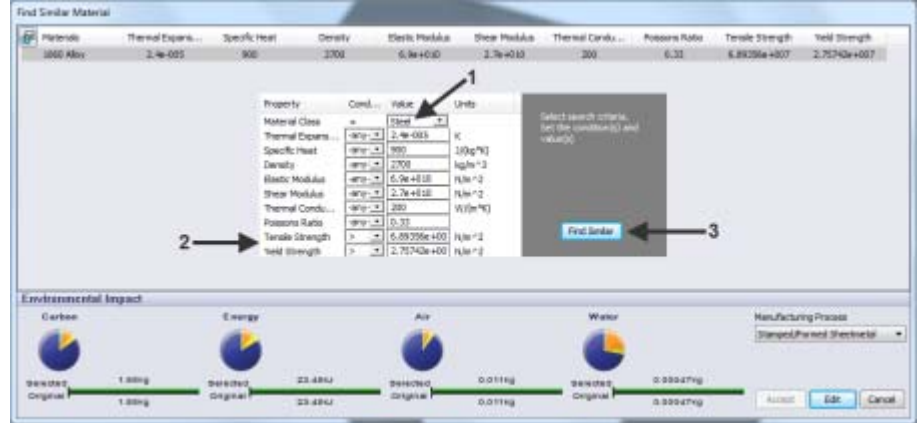
Material Class = **Steel**

2. Let us choose a higher **tensile strength** and a higher **yield value**.

Tensile Strength >
Yield Strength >

3. Click **Find Similar**.

Find Similar



38

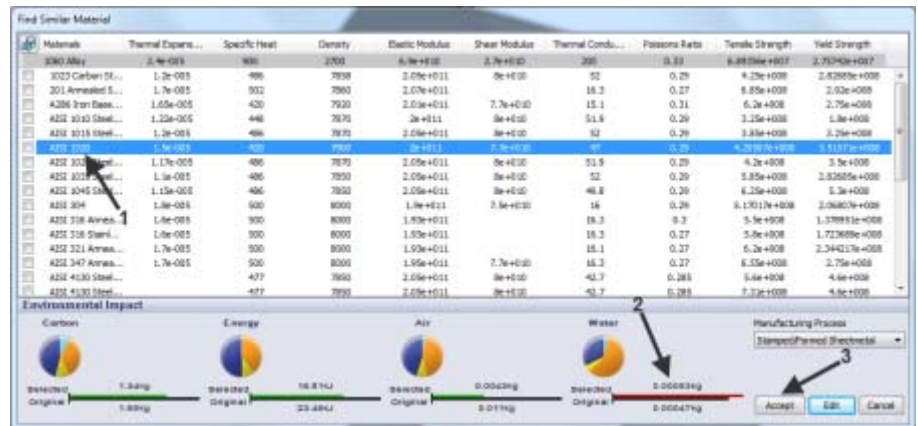
AISI 1020 is a low carbon machine steel offering the good general and structural steel properties. So, we will choose this type of steel.

1. Double-click: **AISI1020**.
2. The diagram section shows this choice is better for the environment, except for the water.



3. Click **Accept**.

Accept

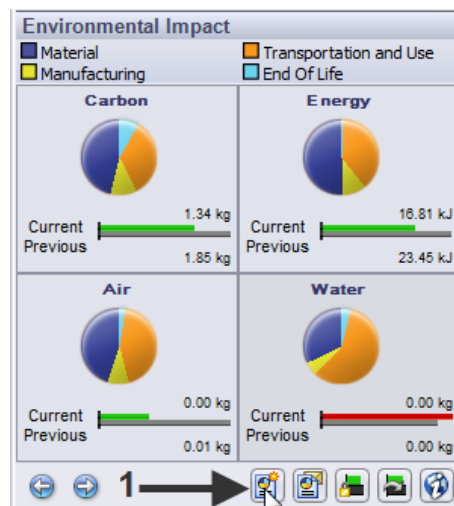



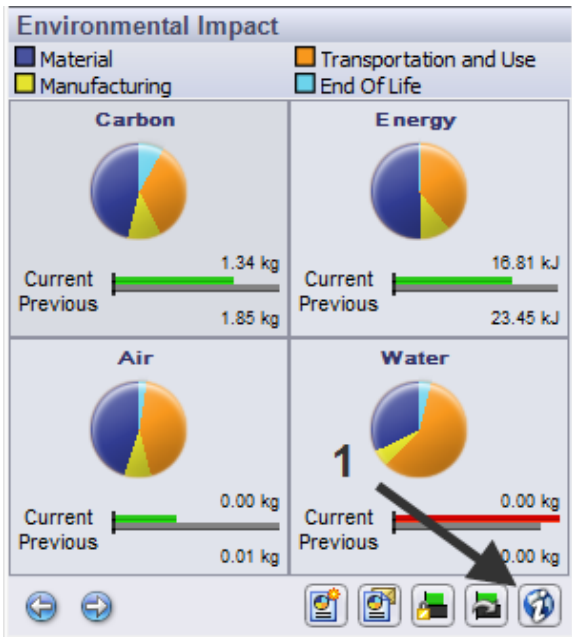


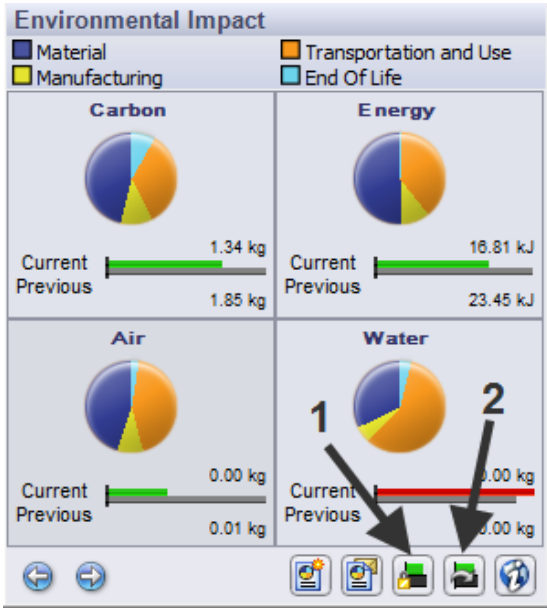
39


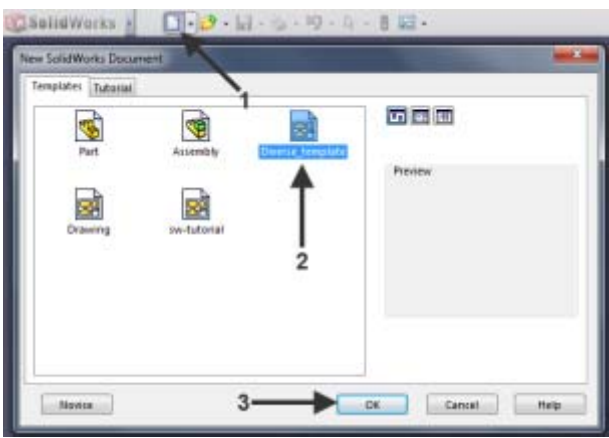
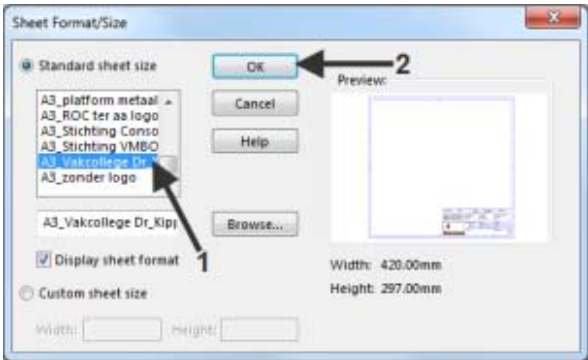
1. Click: **Generate Report**.


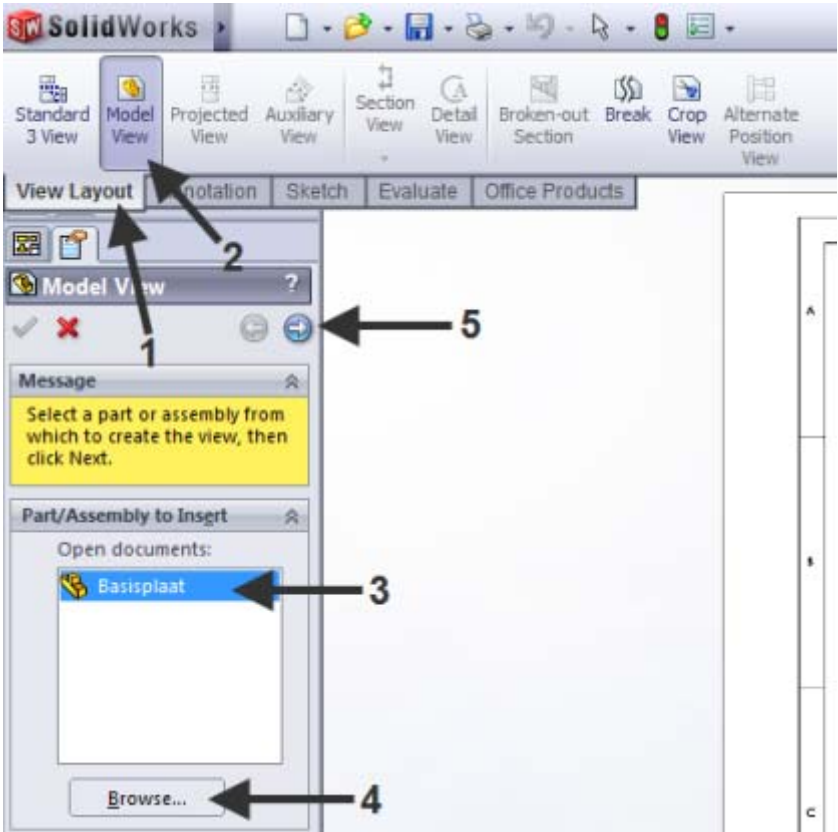
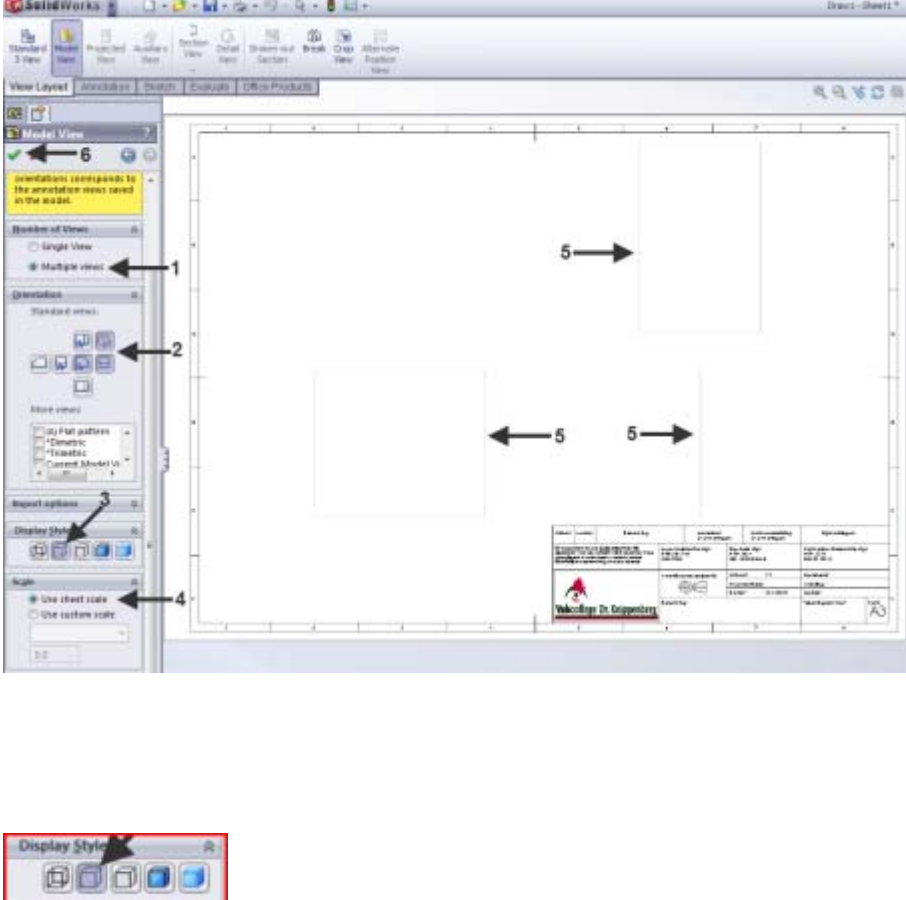




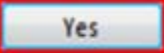
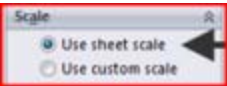
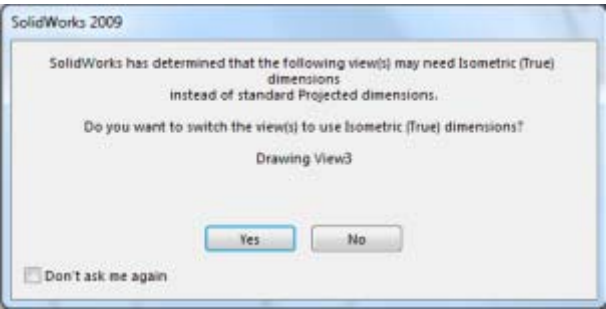


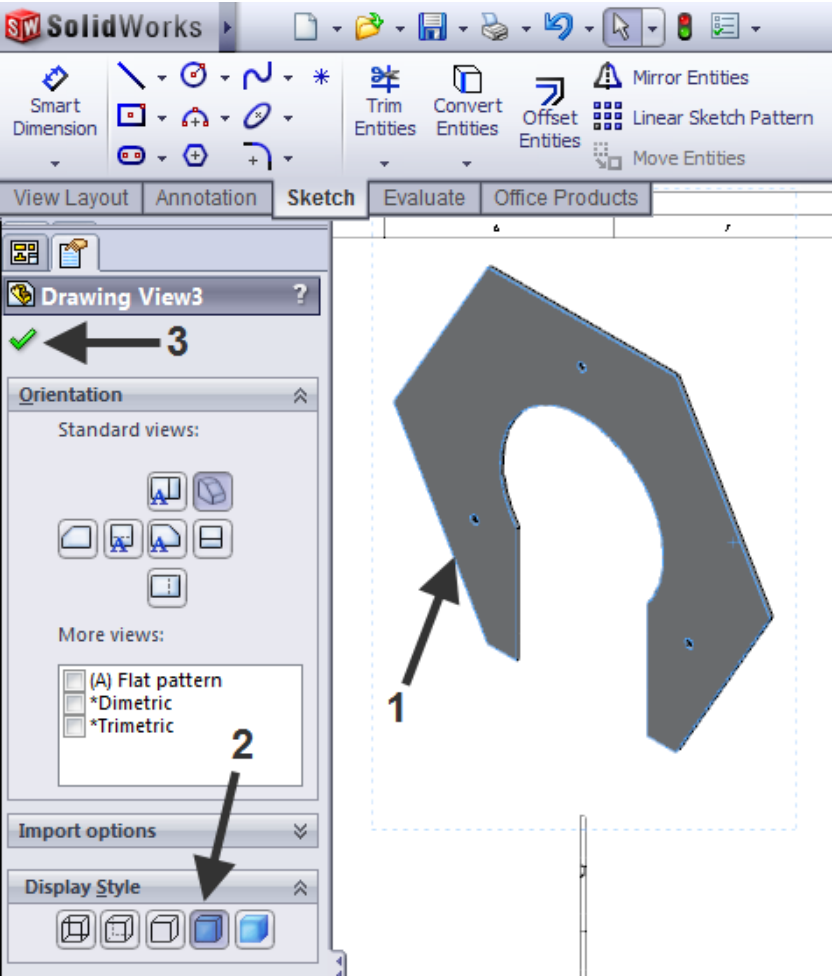
This button allows you to generate a full report.

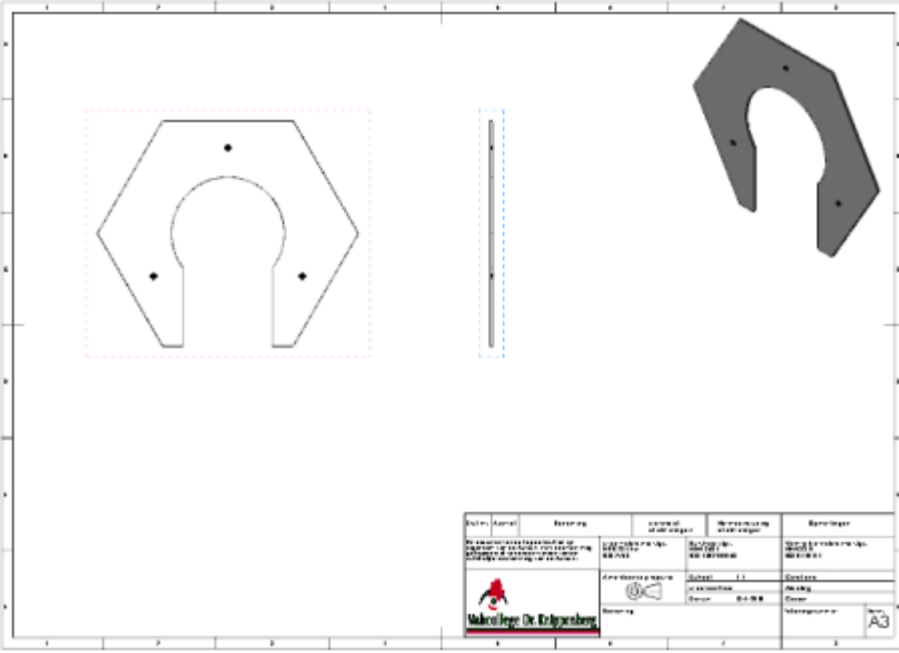
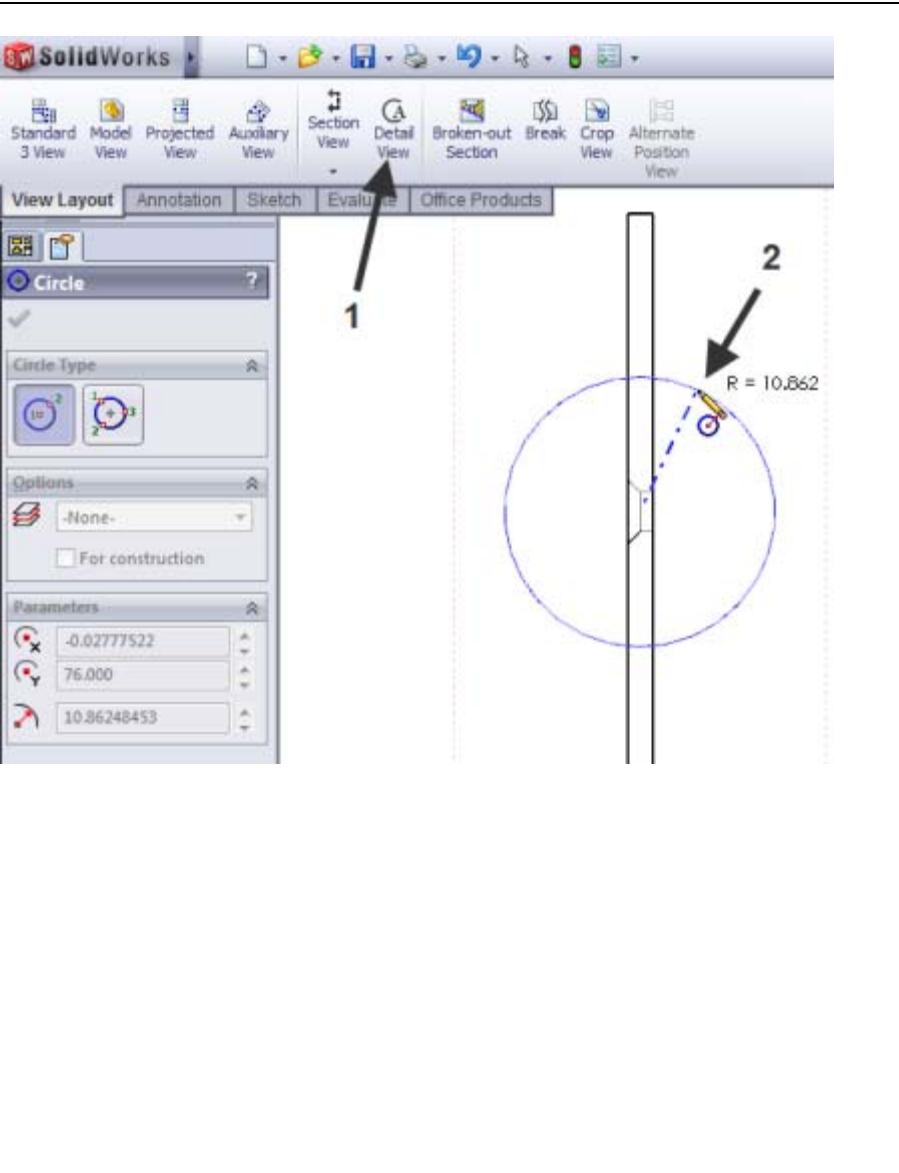


40	<p>The report looks like this. Open the file Sustainability Report.docx to take a closer look.</p> <p>The report allows you to make an informed choice between the original and the alternative material selection.</p>	
41	<p>1. If you click: Online Info, online information on CO2 emission during production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

<p>43</p>	<p>Close the function: Sustainability Xpress.</p> <ol style="list-style-type: none"> 1. Click the red x 2. Now try a few other materials yourself to see which is the best solution (e.g. wood). 	
<p>44</p>	<p>Now, make a drawing for use in the workshop.</p> <ol style="list-style-type: none"> 1. Click New: 2. Select: Diverse_template 3. Click OK. 	
<p>45</p>	<p>In the menu, choose:</p> <ol style="list-style-type: none"> 1. A3_Vakcollege Dr A3 _Vakcollege Dr_Knippenberg <p>Click OK.</p>	

<p>46</p>	<p>An empty drawing field is displayed. Do the following to create views.</p> <ol style="list-style-type: none"> Click View Layout Click Model View to open the Property Manager model view. Make sure the appropriate part has been selected. If not, use the button Browse... to find the appropriate part. Click the arrow  to continue. 	
<p>47</p>	<ol style="list-style-type: none"> In the menu: Number of Views Select: Multiple views. To position three views. In the Orientation menu, choose the front view, the side view and the isometric view. The selected views are displayed in grey. For Display Style, choose Hidden Lines Visible. 	

	<p>4. Leave the scale at: Use sheet scale.</p> <p>5. If all went well, SolidWorks has positioned the three chosen views.</p> <p>6. Click OK. </p> <p></p> <p>Sometimes, a message like in the adjoining figure is displayed.</p> <p>In that case, click: </p>	 
48	<p>1. Now select the isometric view.</p> <p>2. Click:  Shaded With Edges.</p> <p>3. Click OK. </p>	

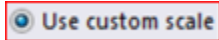
<p>49</p>	<p>Distribute the views on the drawing sheet, as in the adjoining figure.</p>	
<p>50</p>	<p>Add details to the drawing.</p> <ol style="list-style-type: none"> 1. Click Detail View. 2. Draw a circle. 	

51 You will have to enter the following information:

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



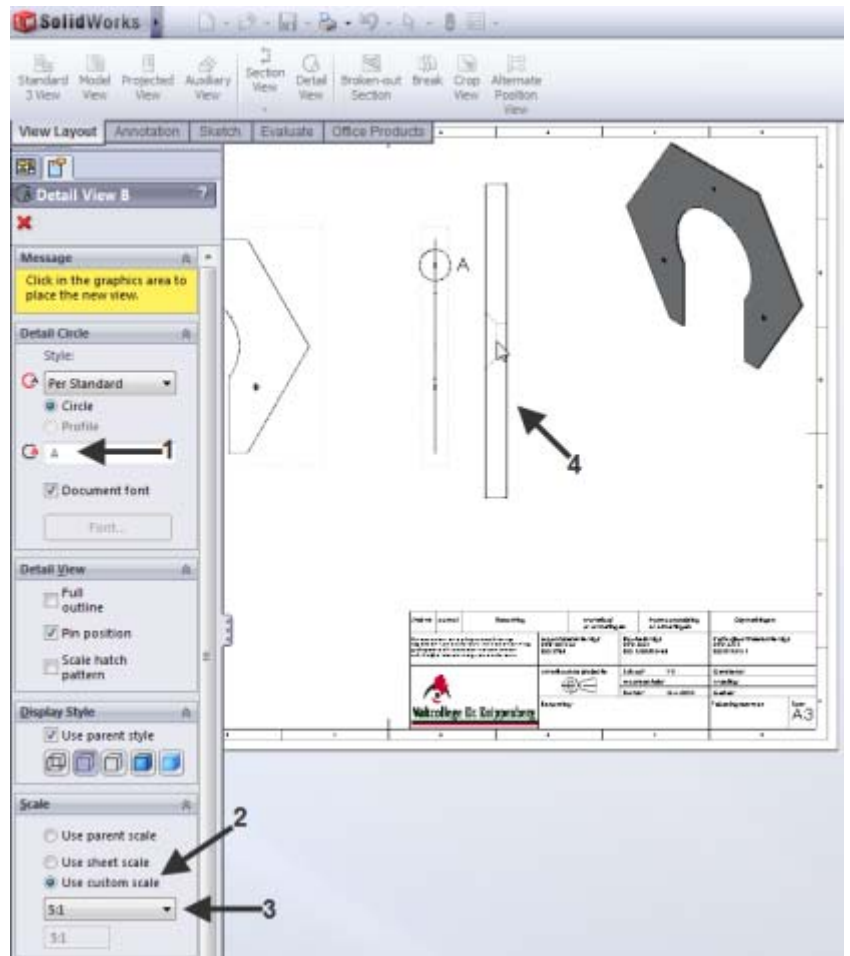
2. We want to enlarge the detail. Therefore, choose:



3. Enter the ratio



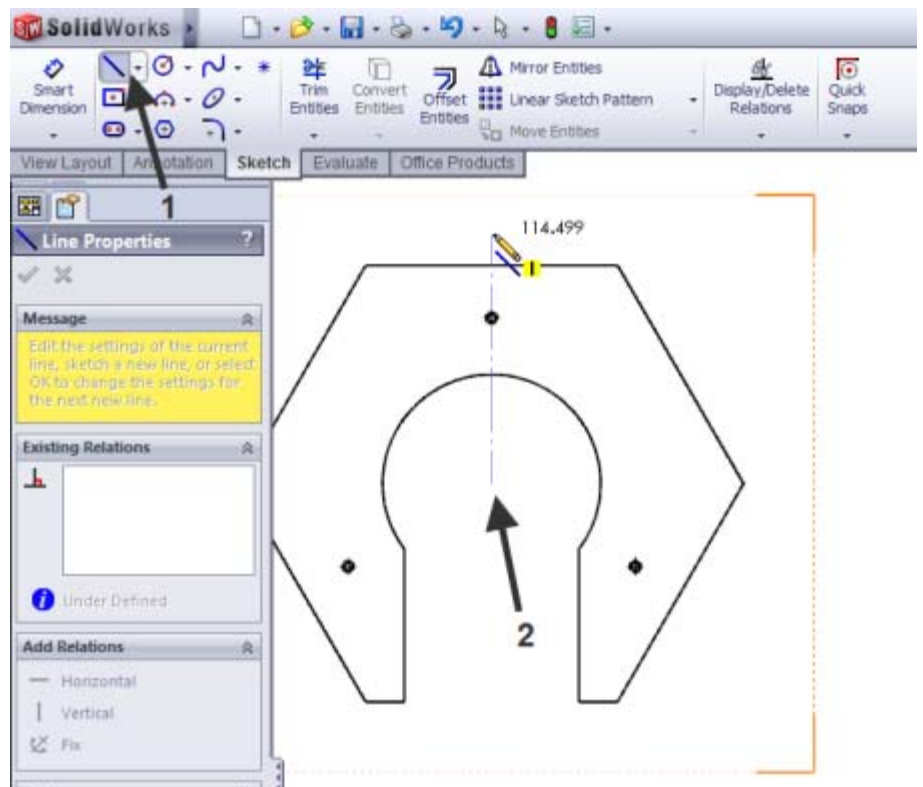
4. Click next to the top view to position the detail.



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



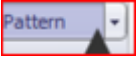
1. Click: the arrow to open the Centerline function and select it.
2. Click in the middle of the circle; draw the Centerline, then click anywhere outside the model.




53

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

1. Click the scroll down

menu  and select:

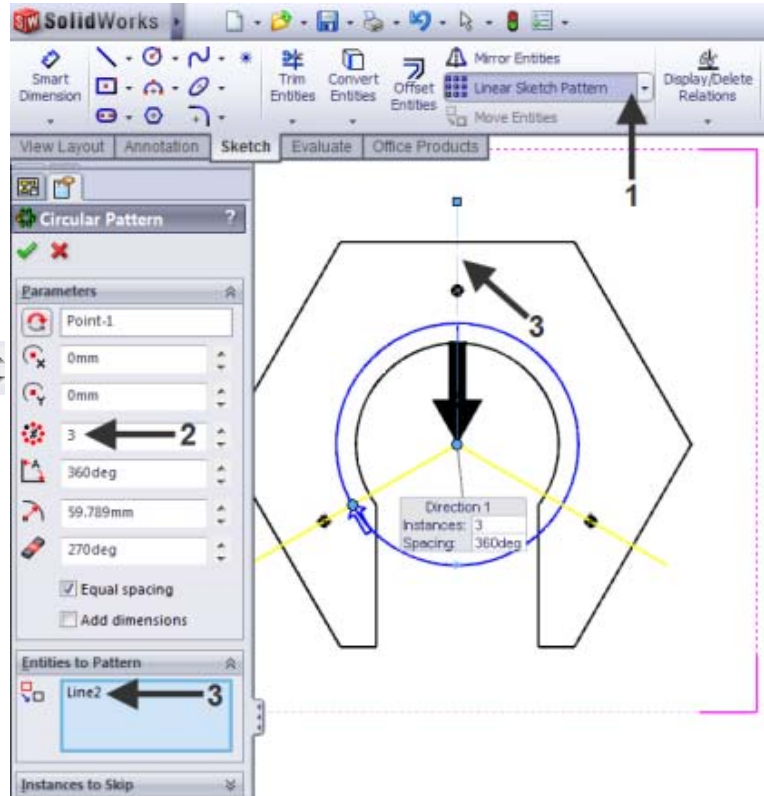
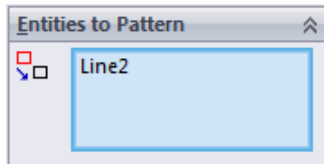
Circular Pattern.

 **Circular Sketch Pattern**

2. Enter 3 as quantity.

 3

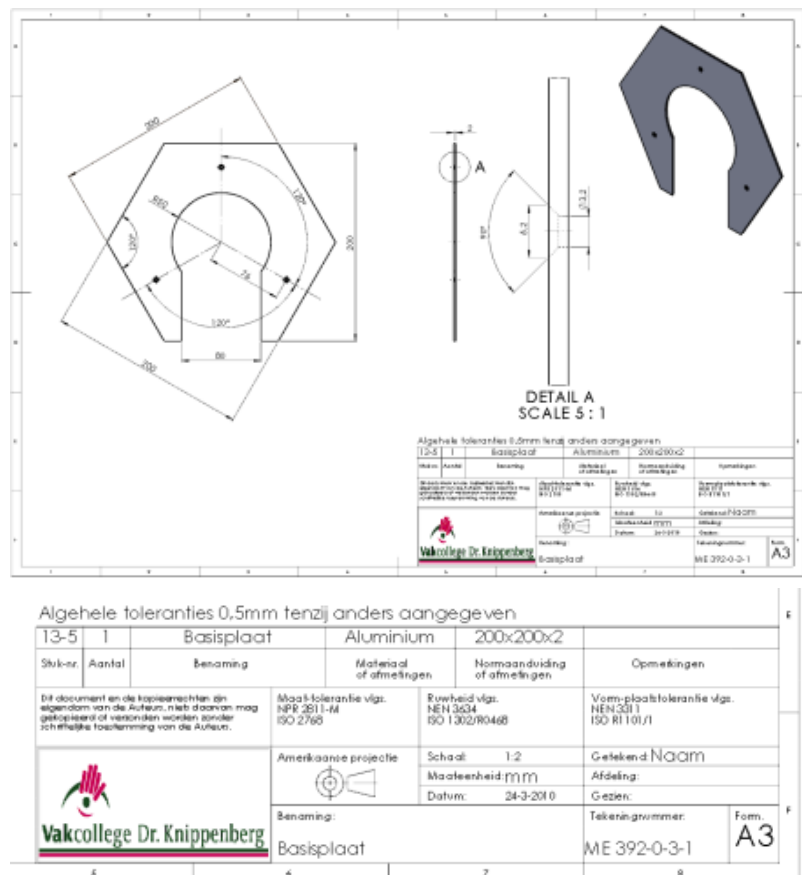
3. Click anywhere in the window: **Entities to Pattern**, then select the vertical line you have just drawn.



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.sldrw



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