SolidWorks® Tutorial 4

CANDLESTICK
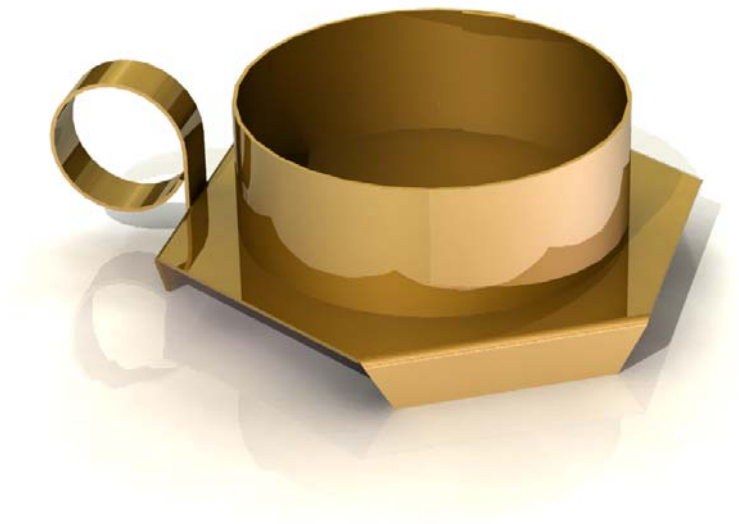
Preparatory Vocational Training
and Advanced Vocational Training

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Candlestick

In this tutorial you will make a simple container and a candlestick out of sheetmetal. You will learn about working with sheet metal in SolidWorks. We will show you a couple of ways to create a product out of sheetmetal and we will show you how to make a drawing in 2D.
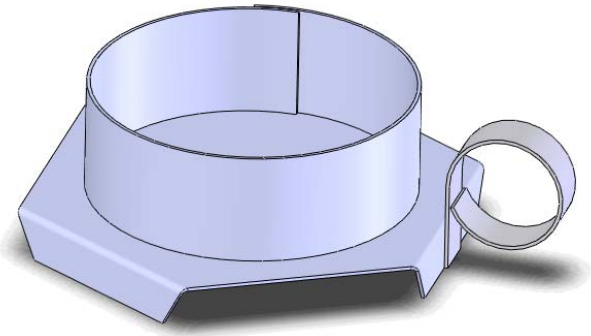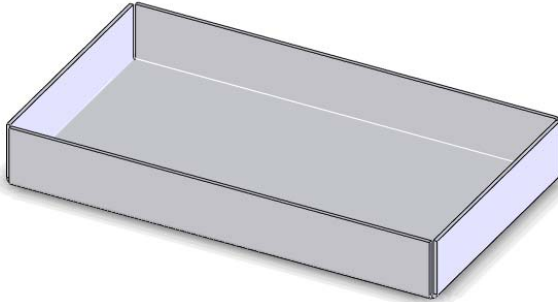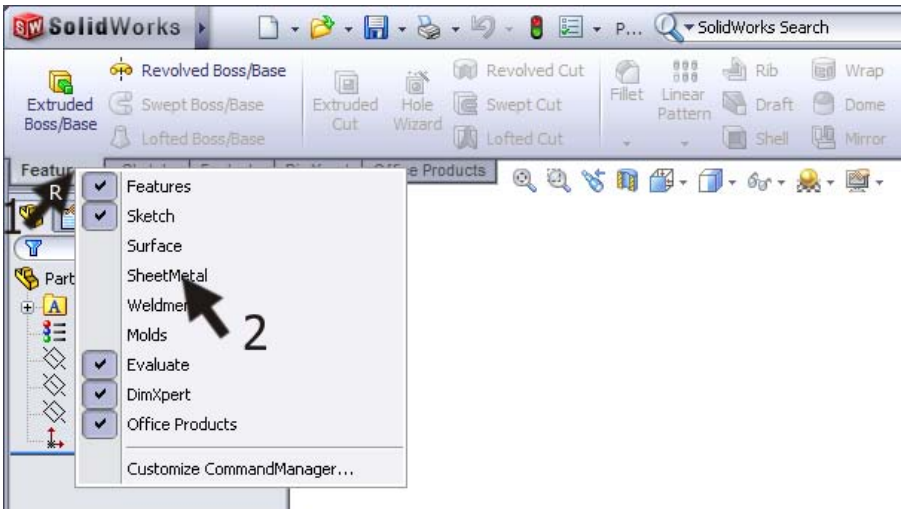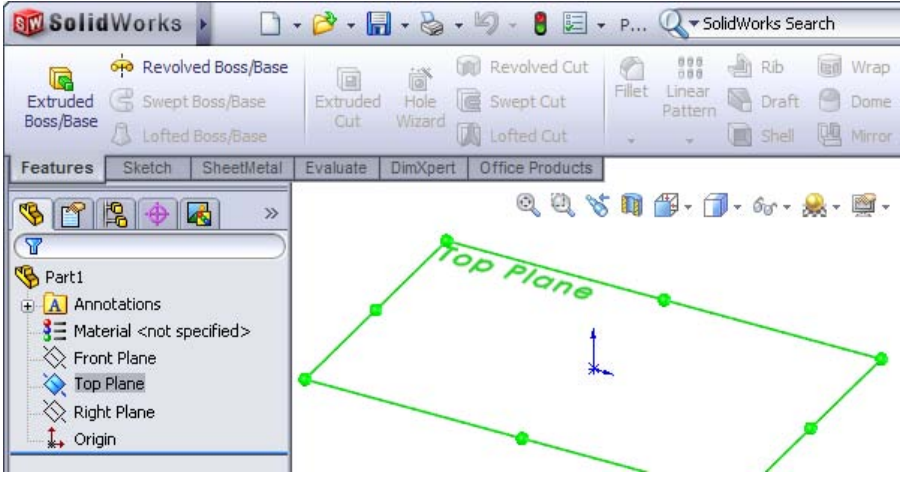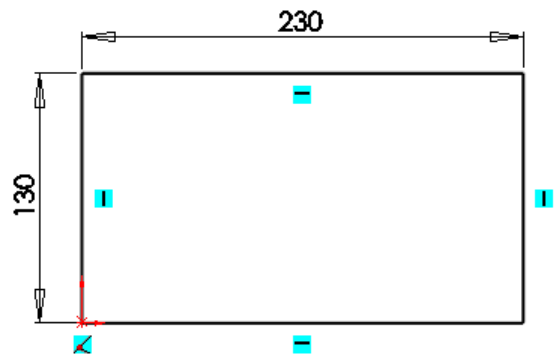
Work plan	First we will make a container. Look at the drawing below.
	230 ... 130 ... 2 ... 30
	We will execute the following steps:
	1. First, we will create the base. For this we will use an outside dimension of 230 x 130.
	2. After that, we will add four sides with a height of 30.
	3. Finally we will look at the 2D drawing of the design.
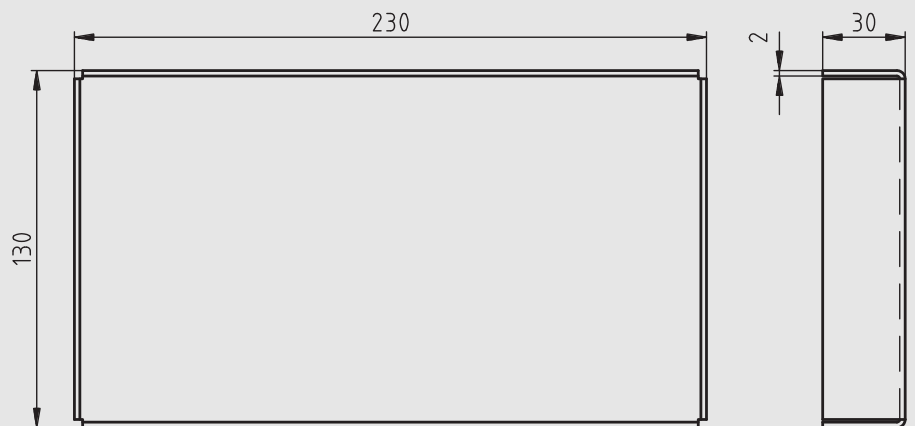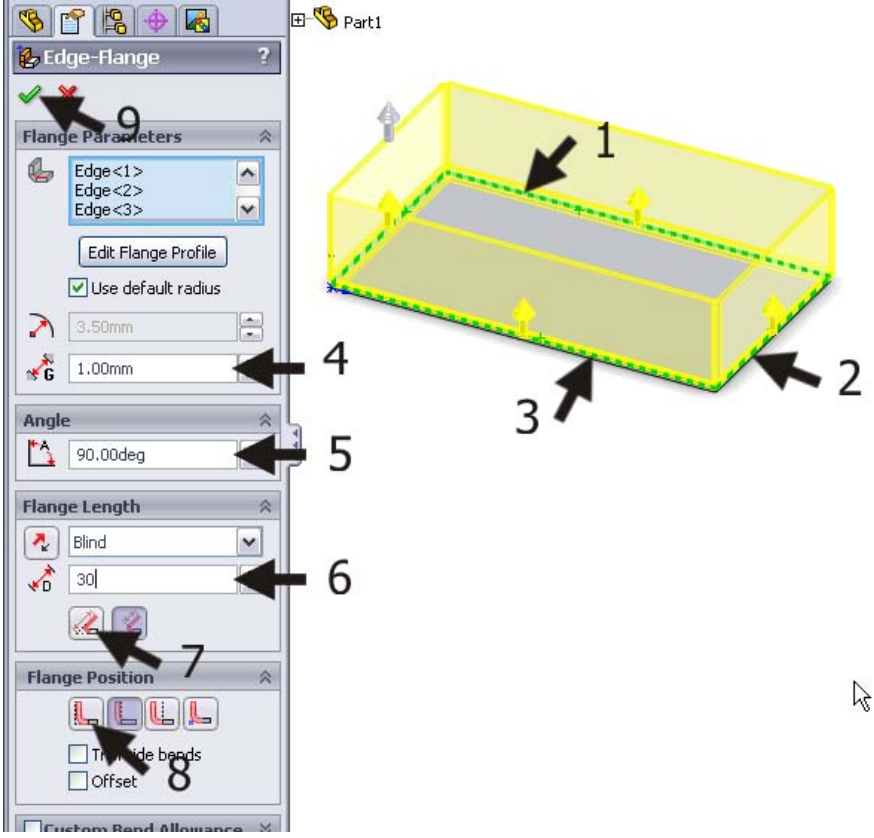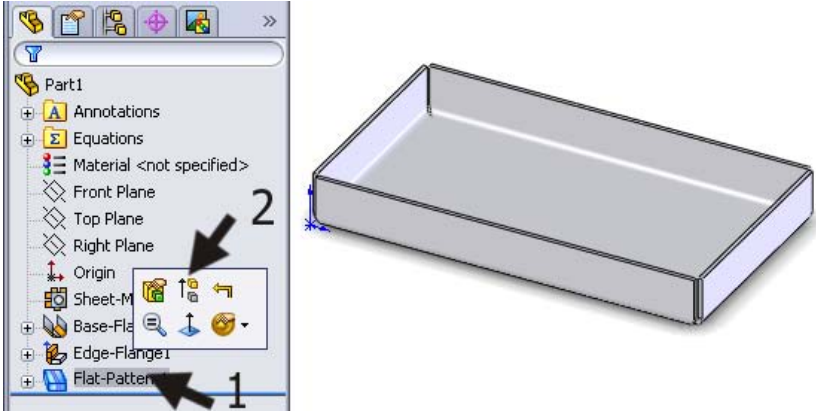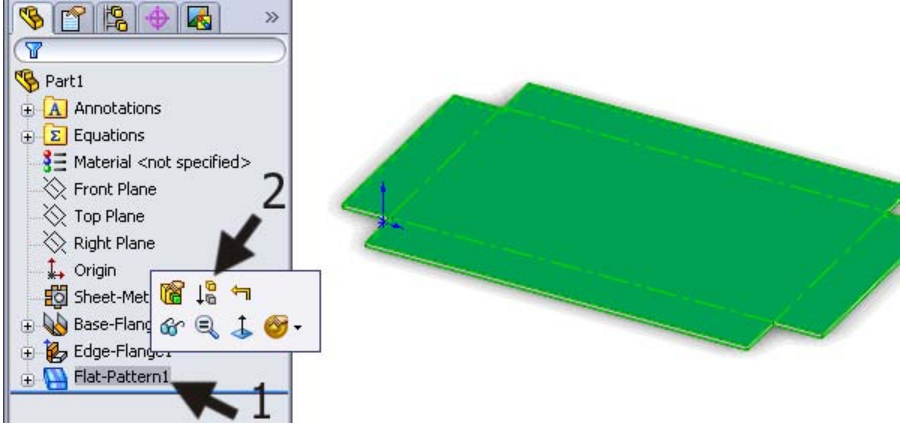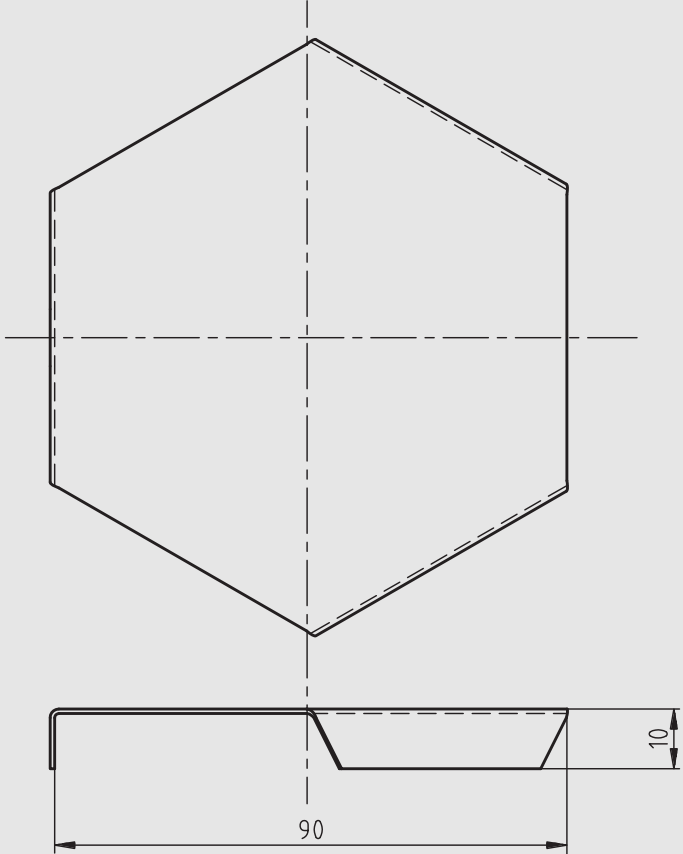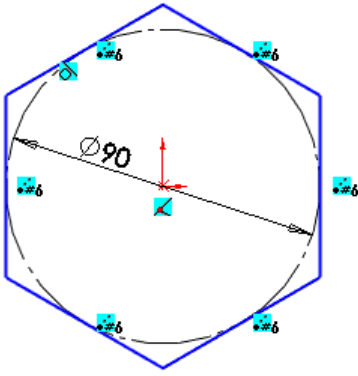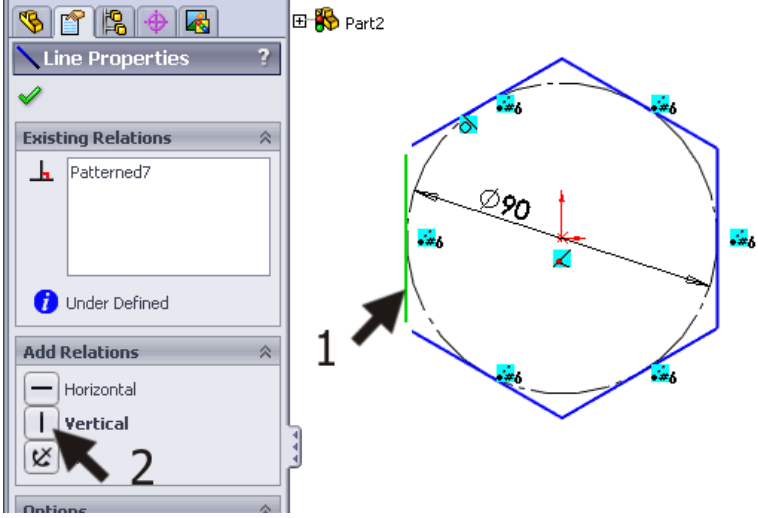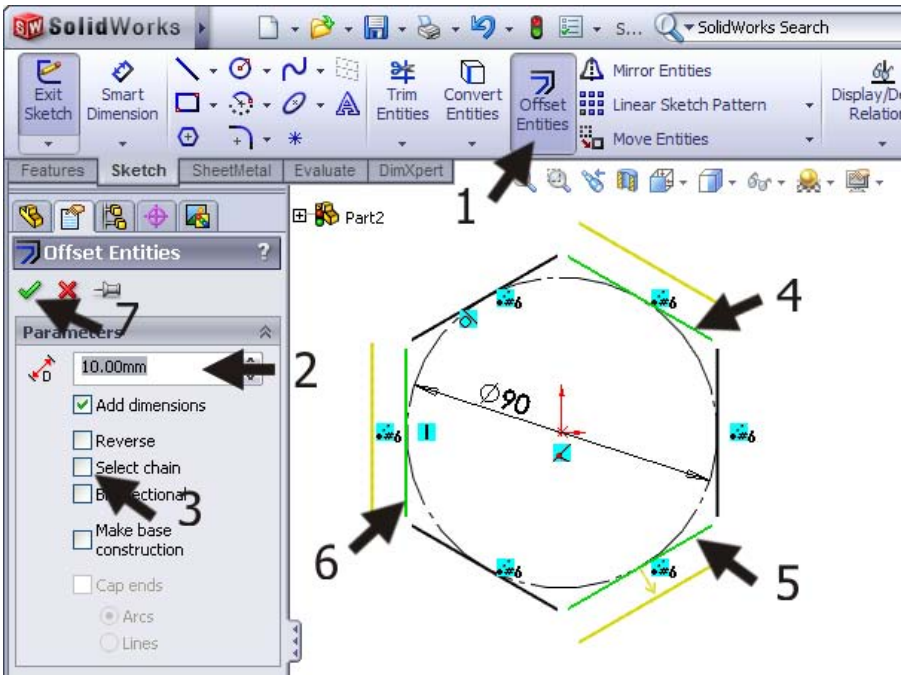
1	Start SolidWorks and open a new part.	
2	Be sure that the buttons you need to work with SheetMetal are visible. The easiest way to access these tools is to add them to the CommandManager. 1 Click on a tab in the CommandManager with the right mouse button. 2 Click on 'SheetMetal' in the menu that appears.	
3	Select 'Top Plane' in the FeatureManager. We will use this plane to create a sketch.	
4	Create the sketch like in the illustration on the right. Draw a rectangle with one corner above the origin. Set the dimension of the height to 130 and the width to 230. Do you still remember how to start a sketch? If not, look at step 2 and 3 of Tutorial #3.	
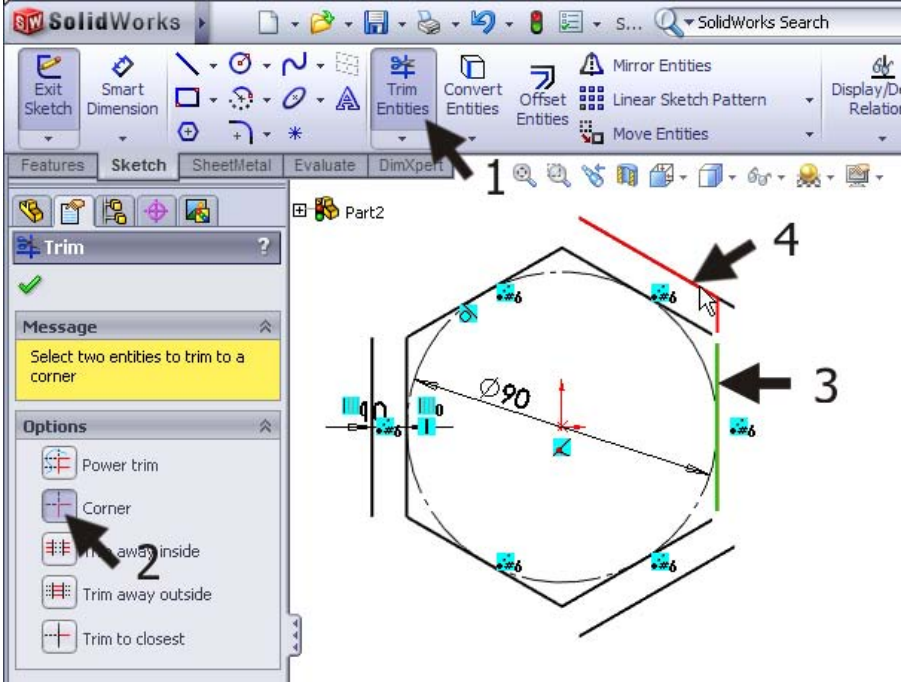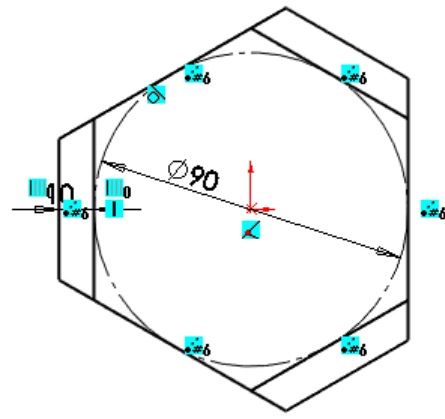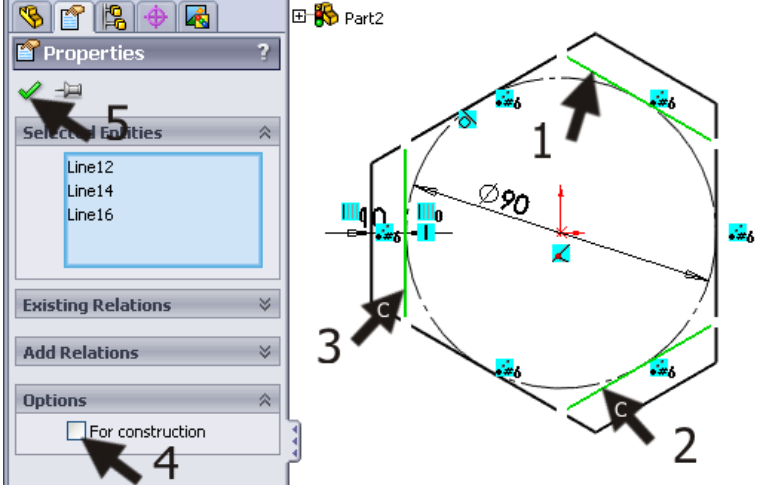
5	Next, click on 'SheetMetal' in the CommandManager and then next on 'Base Flange'.	
6	1. Set the thickness at 2 mm in the Property-Manager. 2. Click on OK.	
7	To create the edges of the container, click on 'Edge-Flange' in the Command-Manager.	
8	1 Click on the first edge of the base and move the mouse upwards. 2 Set the first rim with a random height.	

| 9 | 1-3 Next, click on the other edges. Their heights will automatically adjust to the first one.

Change a few settings in the PropertyManager as is shown in the illustration at right:

4. Set the gap between the walls at 1mm.

5. The walls are at a 90° angle to the base.

6. The height of the walls is 30mm.

7. This height is measured from the outside of the base.

8. The walls are placed within the outside edge from the base and on top of the base.

9. When the settings are correct, click on OK. | |
|---|---|---|
| 10 | The container is ready now.

Next we will make a 2D drawing.

The last feature in the FeatureManager is 'Flat-Pattern1'. Normally this feature is suppressed (gray-colored in the FeatureManager), and we have a normal view of the model.

By setting this feature to unsuppressed, we will get a 2D drawing of the model.

1. Click on the last feature in the FeatureManager,

2. Select Unsuppress in the menu. | |
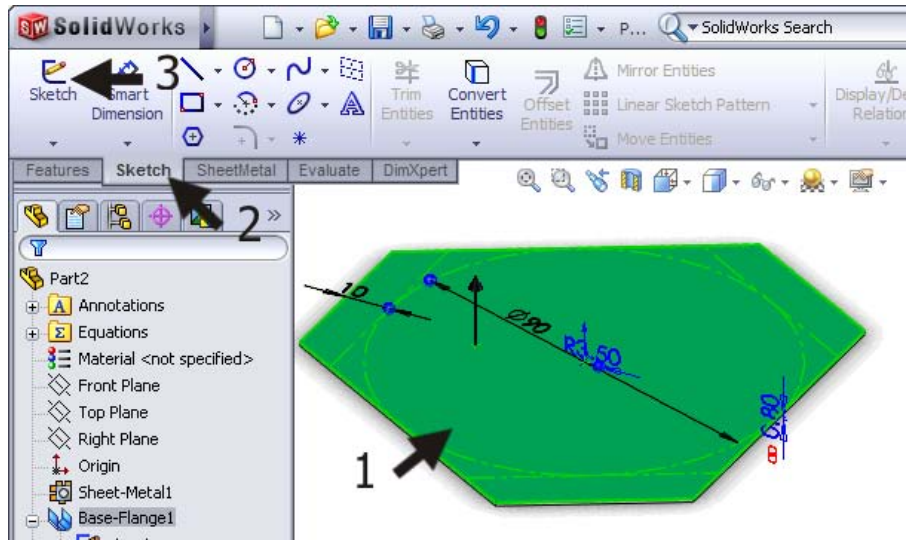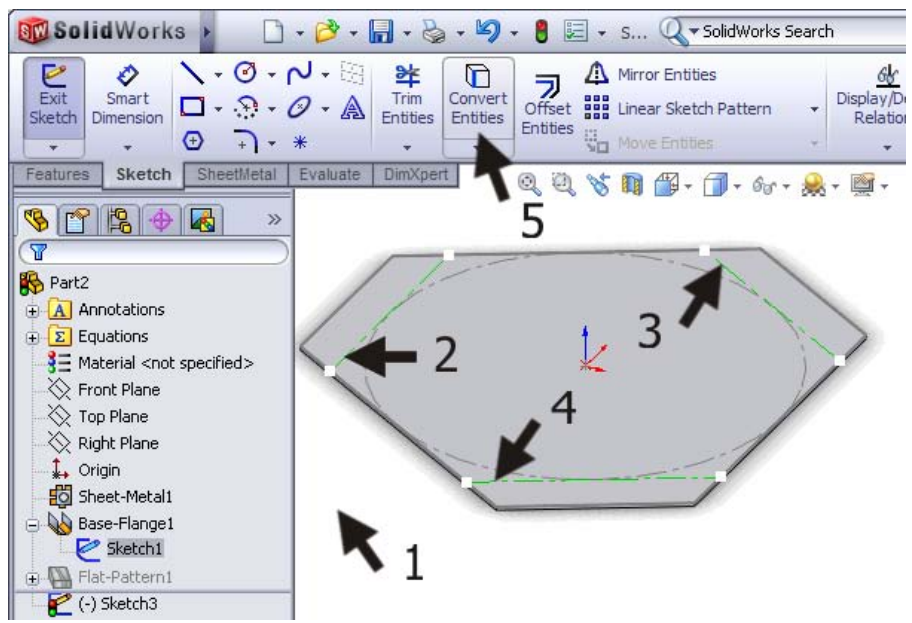
11	At this point, a 2D drawing of the container is visible. If you want to return to the normal view in 3D, click on the last feature again and select Suppress.	
12	Save the model as: box.SLDPRT.	
	Work plan	We are going to create a candlestick. It consists of three parts. First, we will create the base in accordance with the drawing below. We will handle this product differently than we handled the others. We draw a 2D drawing and bring in some bending lines. The hardest part of this model is to make the first sketch.
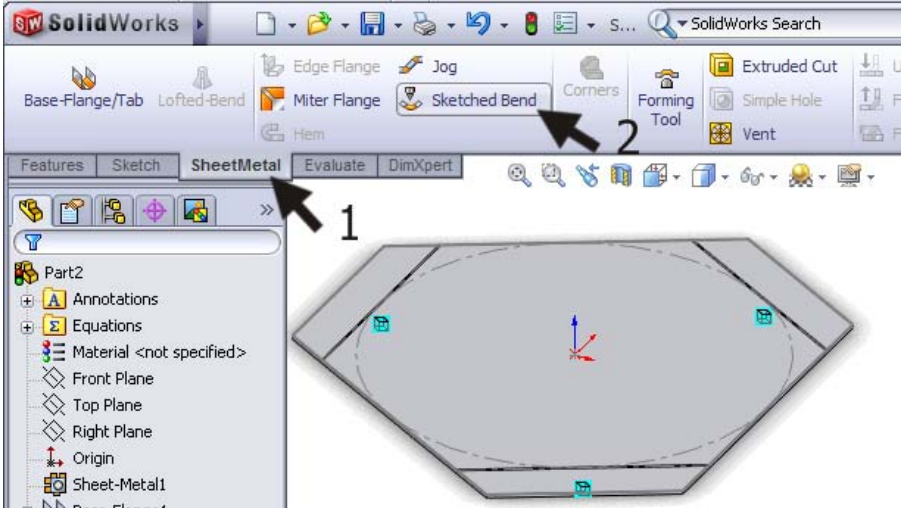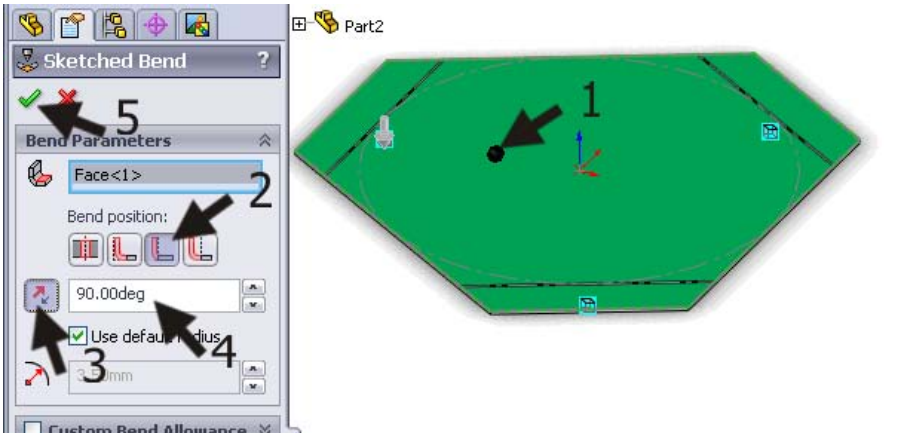
13	Open a new part.	
14	1. Select the 'Top Plane', to make a sketch on it. 2. Click on Polygon in the CommandManager.	
15	Click on the origin for the first dot of the hexagon and at a point straight above the origin at a random distance from the first one.	
16	Be sure that in the PropertyManager: 1. The number of sides is set to 6. 2. The dimension of the inner circle is set. 3. Click on OK.	
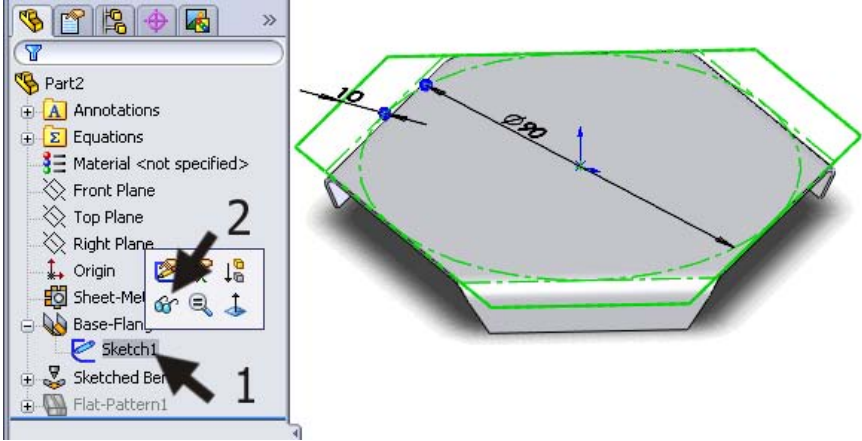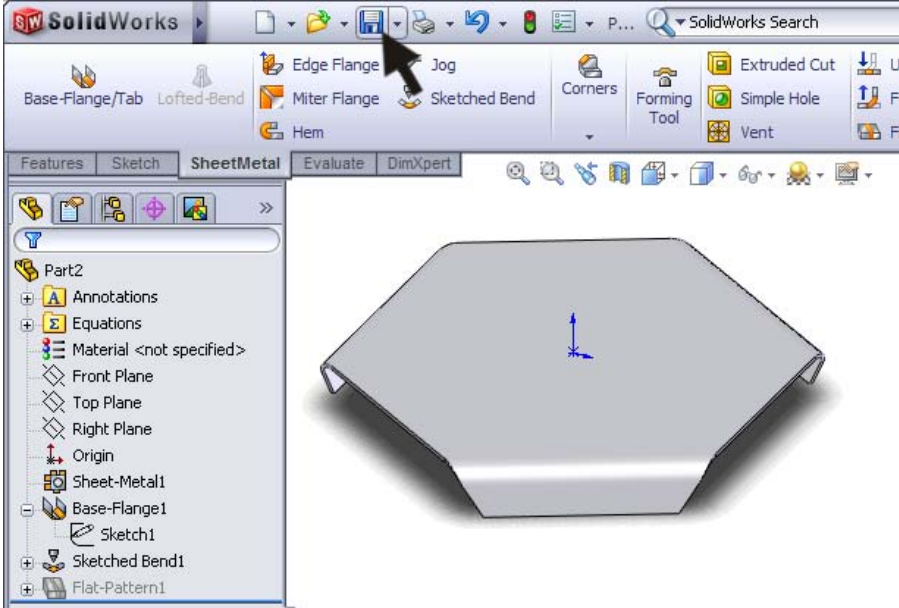
17	Set the dimension of the inner circle to 90 mm with Smart Dimension.	
18	To set the direction of the hexagon you do as described below: 1. Select ONE of the vertical sides of the hexagon. 2. Click on Vertical in the PropertyManager.	
19	1 Click on 'Offset Entities' in the CommandManager. 2 Set the distance in the PropertyManager to 10 mm. 3 Copy the other settings of the PropertyManager from the drawing at the right. Be sure the option 'Select Chain' is NOT selected. 4-6 Select the sides of the hexagon as shown at right. Pay Attention: when the lines are off-set to the inside, check the option 'Reverse' in the PropertyManager. 7. Click on OK.	
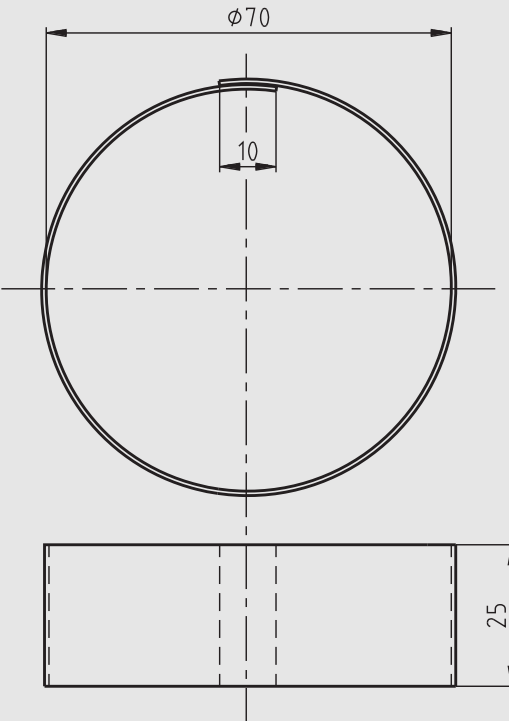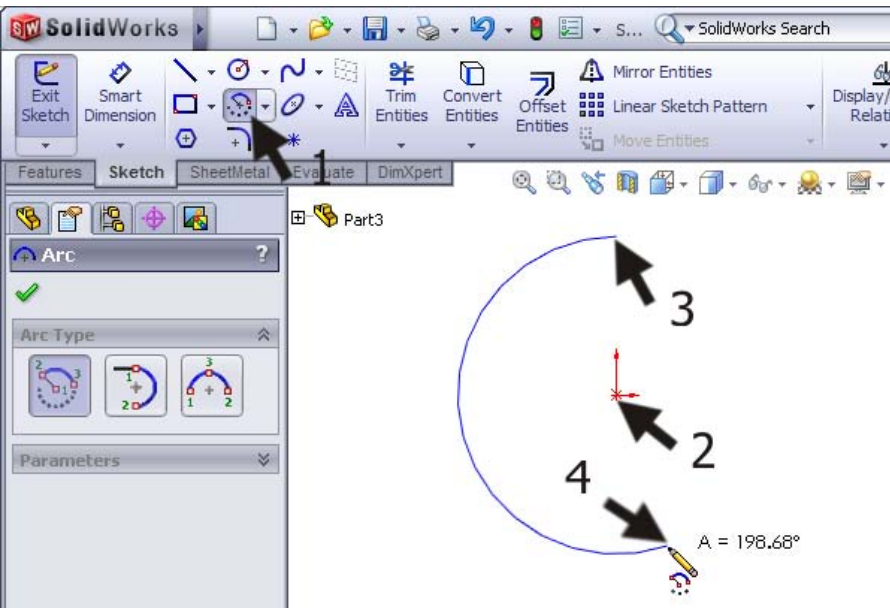
20	1. Click on 'Trim Entities' in the CommandManager. 2. Select the option Corner in the PropertyManager. 3-4 Click in the sketch on two lines that form a corner.	
21	Click two lines again and again so you see the drawing as shown at right.	
22	Finally, we will transform the three inner lines into construction lines. This will create the bending lines we will use later on. 1-3 Select the three lines (use the <Ctrl> button on your keyboard). 4. Check the option For construction in the PropertyManager. 5. Click on OK.	

23	Next, create the base. 1. Click on 'SheetMetal' in the CommandManager. 2. Click on 'Base-Flange'.	
24	1. Set the thickness of the material to 0.8 mm in the PropertyManager. 2. Be sure to check or uncheck the option 'Reverse direction' to add the material at the bottom of the base material. Do you have a good view at the material? When not, zoom in! 3. Click on OK.	
25	In the sketch we have just created, the bending lines have already been drawn. We are going to use them now, but for this purpose, the sketch must be visible. 1 Click on the '+' sign in front of 'Base-Flange1' in the FeatureManager. 2 Now, click on the sketch that is visible (usually this is: 'Sketch1'). 3 Click on 'Show' in the menu that appears. The sketch is now gray-colored in the model.	
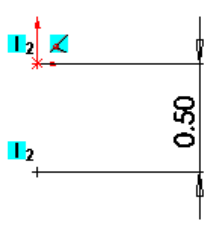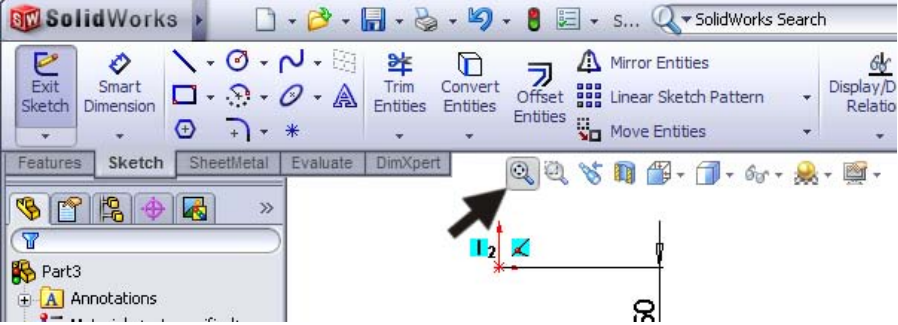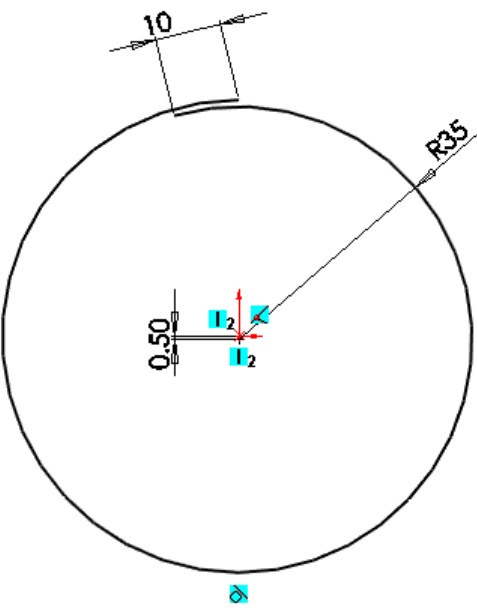
26	Start a new sketch at the top plane: 1. Select the top plane of the item you have just created 2. Click on 'Sketch' in the CommandManager to show the right buttons. 3. Click on the 'Sketch' command to open the sketch.	
	Tip!	In earlier exercises, we opened a sketch by selecting a plane and drawing a rectangle (example). SolidWorks 'understands' that in such a case you want to open a sketch and does so automatically. Before you can use the command for the next step, a sketch **must be** open already; otherwise the command will not be visible. For this reason, we must open the sketch ourselves and that is exactly what we have done in the last step.
27	1. Click somewhere beside the model to unselect the plane. 2-4 Select the three bending lines from the last sketch. Use the <Ctrl> button. 5. Click on 'Convert Entities' in the Command-Manager.	
	Tip!	For a lot of features in SolidWorks, you must first make a sketch. So you cannot use an edge or an existing line to use them in a new feature. But you CAN do what we have just done here: make a copy of an existing element and paste it in a new sketch. This can be a line from an old sketch but it can also be an edge of a model or even a face. In this way, you can make a new sketch that is derived from the existing model. When an element is not exactly in the plane of the sketch, it will be projected on it.

29	1. Click on 'SheetMetal' in the CommandManager. 2. Click on 'Sketched Bend'.	
29	Unfortunately, this function has no preview. You have to set a number of elements without seeing the exact results. 1 Click at a position in the middle of the base to confirm which part of the base is fixed. We will bend the other parts later on. 2 Select the option Material outside: this is related to the way in which the dimensions are in the drawing. 3 With the Reverse direction button you determine in which direction the material is bent (up or down), and the arrow gives you the direction and can be changed by clicking on this button. Make sure the arrow points downwards. 4 Set the corner at 90°. 5 Click on OK.	
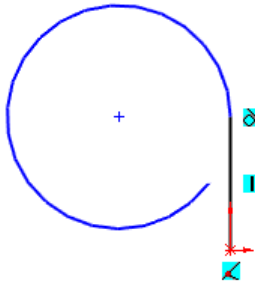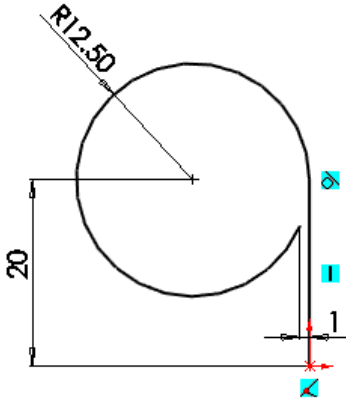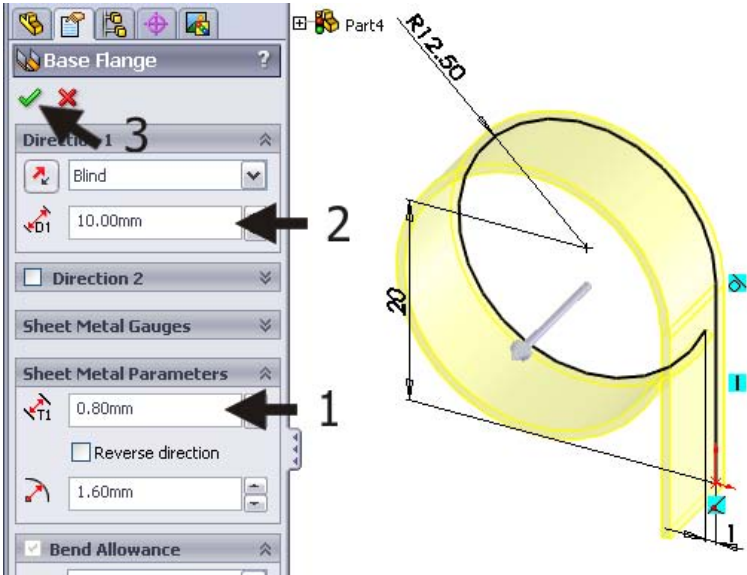
30	Finally, we will hide the sketch we have revealed earlier. Click on the sketch, and select Hide.	
31	The model is ready now. Save is as base.SLDPRT.	

	Work plan	The second part of the candlestick is the 'tube' to put the candle in. This is shaped from a piece of sheetmetal as shown in the drawing below.
		To make this part, we only have to make one sketch.
32	Open a new part and select Top Plane to create a sketch.	
33	First, we will draw one half of a circle. 1. Click on Centerpoint Arc in the Command-Manager. 2. Click on the origin for the first point. 3. Click directly above the origin to get a second point. 4. To finish this half, click on a third point, directly below the origin.	
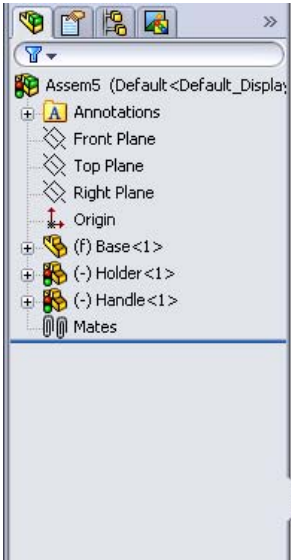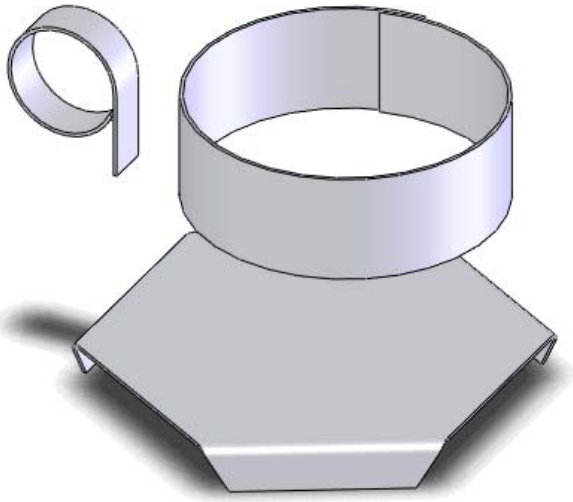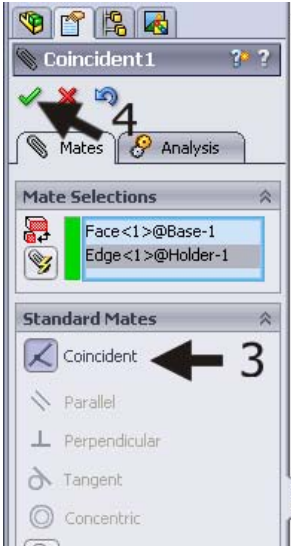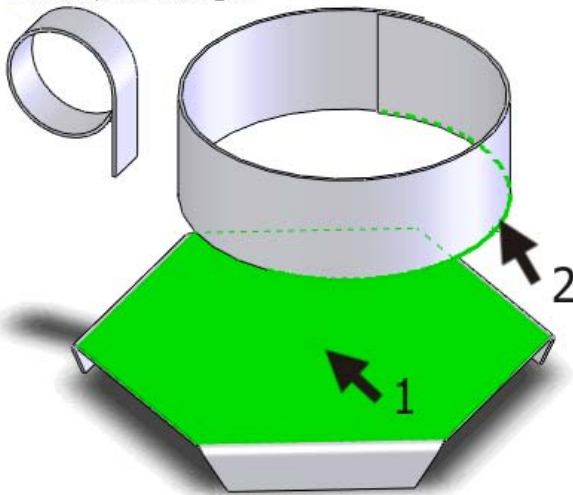
| 34 | Next, we will draw the second part of the circle.

1. Click on Tangent Arc in the PropertyManager.

2. Click on the bottom point of the arc you just drew first.

3. Click on a point as shown in the illustration.

4. Stop the command by pushing the <Esc> button. |
|----|----|
| 35 | Zoom in on the origin of the circle with the center of the second circle also visible. The last one is marked with a little blue '+' mark.

To zoom in, use the scroll wheel of the mouse OR click on Zoom to Area in the View Toolbar. |
| 36 | Select both points and click on 'Vertical' in the PropertyManager. |
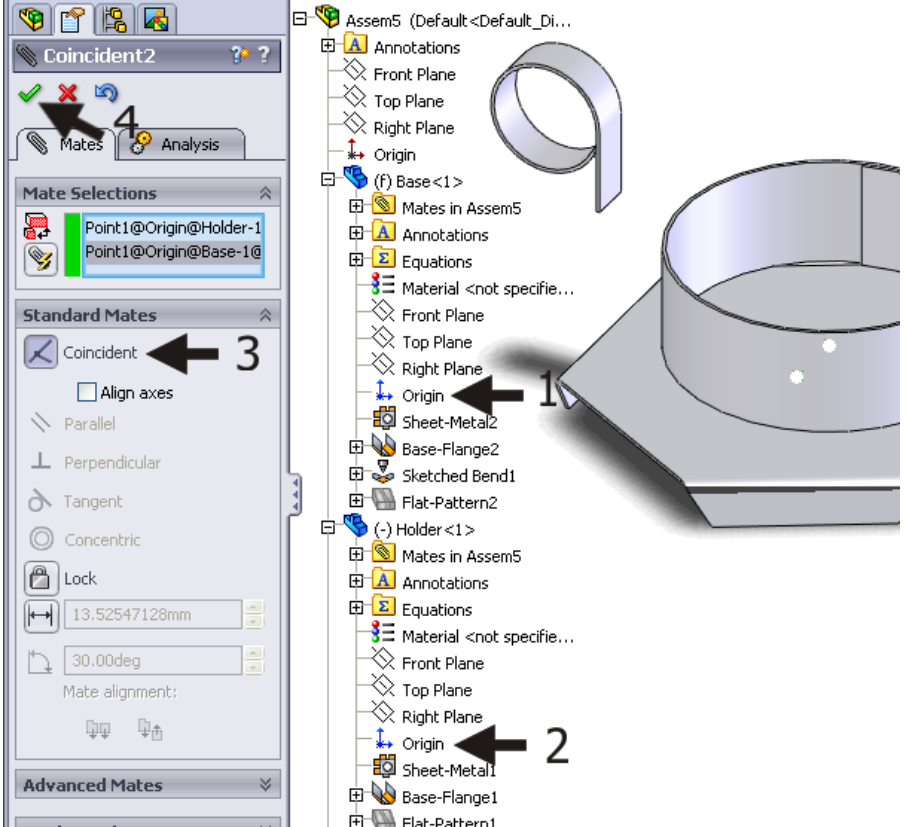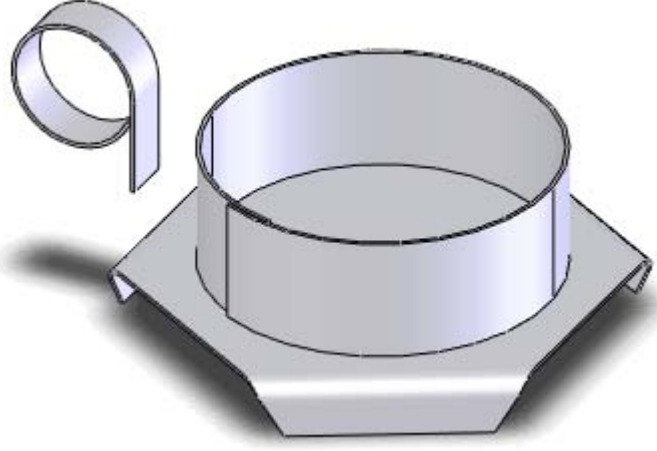
37	Next, set a dimension of 0.5mm between both points.	
38	Next, click on Zoom to fit in the View Toolbar to show the entire sketch.	
39	Add two more dimensions tot the sketch with the Smart Dimension command: 1. A radius of 35 for the right arc. 2. A length of 10mm for the overlap between the first circle and the second one. Pay attention: use the real distance between the ends of the circles and NOT the horizontal distance. This is determined when you set the dimension.	
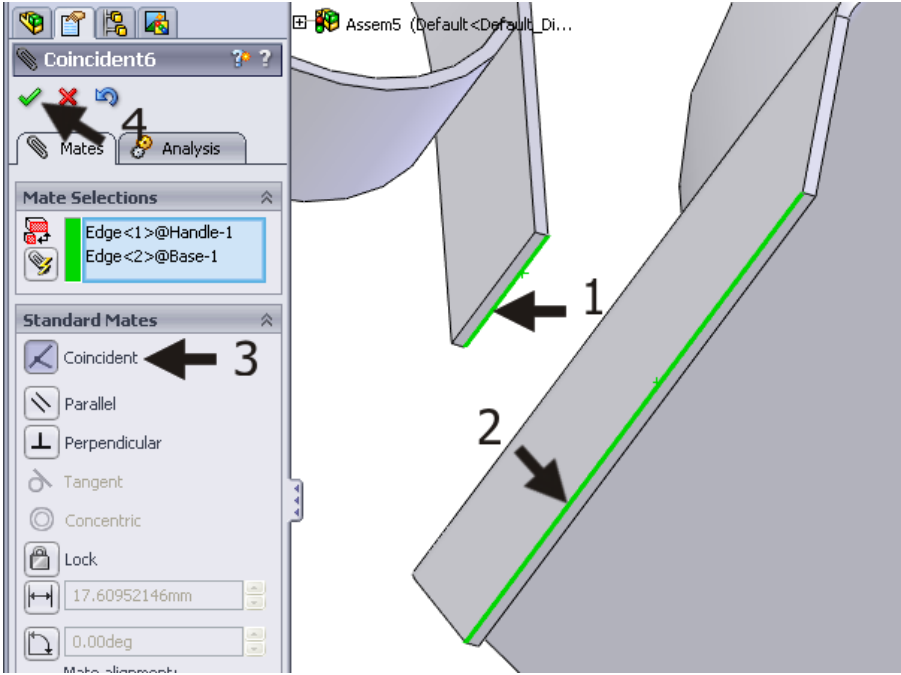
40	Click on 'SheetMetal' in the CommandManager and next on 'Base Flange'. Set the following features in the PropertyManager: 1. Thickness of the material 0.8mm. 2. Height to 25mm. 3. Click on OK.	
41	The cylinder is ready now. Save the file as holder.SLDPRT.	
	Work plan	Finally we have to make the 'ear' of the candle stick. This is done using the same method we used for the last part. Again, the most important step is making a sketch.
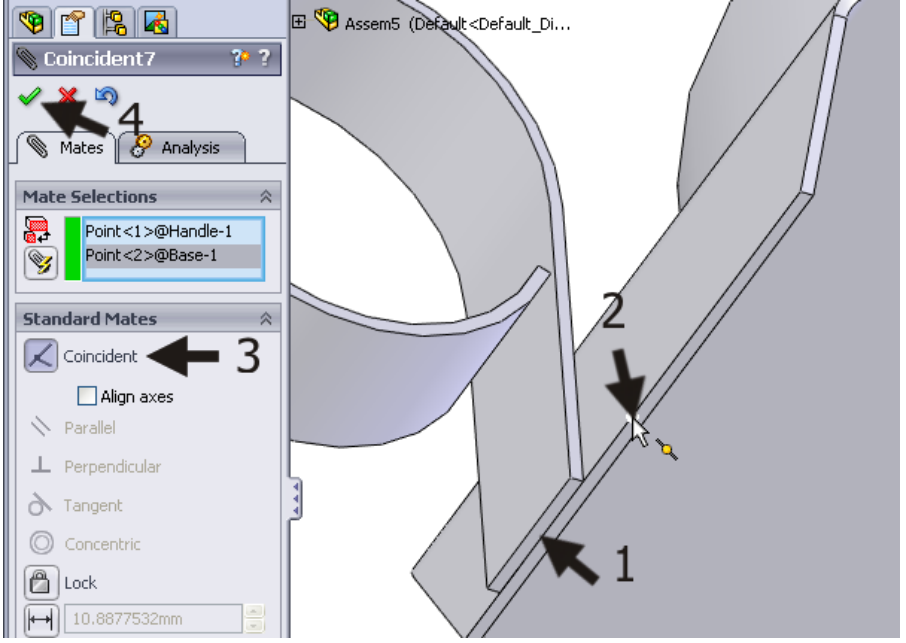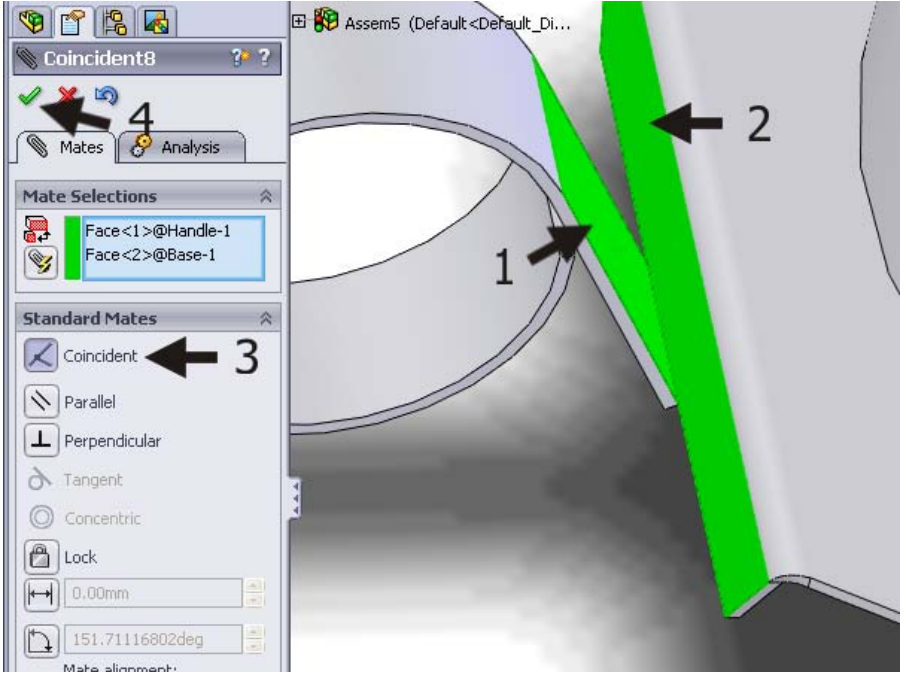
42	Open a new part and start drawing a sketch at the Front Plane. Draw a line from the origin up. Use the Tangent Arc command to draw a part of a circle (an arc) as is shown in the illustration.	
43	Add three dimensions with Smart Dimension as in the illustration on the right.	
44	Use the 'Base Flange' command to set the thickness of the material to 0.8mm and a height of 10mm.	

| 45 | Save the file as handle.SLDPRT. | |
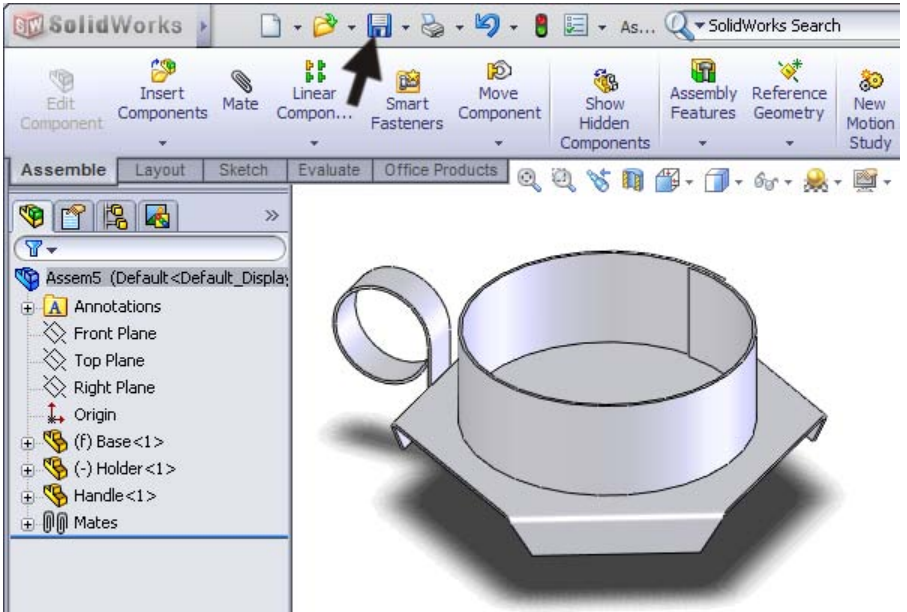
		At the end of this tutorial we will make an assembly. We have done this before. Would you be able to join the three parts together in an assembly? Try it yourself first, before you continue with this tutorial!
46	Open a new assembly. Use the Insert Components command to place the base in the assembly. This will be Fixed. After that, put the two other parts at a random position in the drawing field. Can you remember how this is done? If not, check Tutorial 3 steps 47 to 51.	
47	We have to mate the parts together. Click on Mate in the CommandManager. 1. Select the top plane of the base. 2. Select the bottom edge of the holder. 3. The mate type 'Coincident' is selected automatically. 4. Click on OK.	
Tip!		When your first Mate is finished, click on OK. The Mate command will remain active. You can immediately select two other elements to mate. When you click on OK twice, the Mate command will end. SolidWorks assumes that you want to stay within the Mate command. If you click twice on OK by accident, click on the Mate command in the CommandManager to start a new Mate.

48	Be sure the Mate-command is active (read the tip above). 1. Select the origin of the base in the Feature Tree. 2. Also select the origin of the holder. 3. The mate type 'Coincident' is again selected automatically. 4. Click on OK.	
49	Be sure the 'ear' is placed in the area where it has to be at the end. Look at the illustration at right. When this part is placed somewhere else, you can drag it to its correct position.	
	Tip!	We are using illustrations of the model in which the model is rotated in such a way that either edges or points that are needed to create a mate remain visible at the same time. This is the most convenient approach, because there will be no need to rotate the model during mating. If this does not work, you will have to rotate the model during the mating command like this: 1. Select the first element. 2. Rotate the model so you can get a good view at the second element. 3. Select the second element. 4. Create the mate.

		During this process, be sure not to close the mate command by accident. So pay attention and focus!
50	Rotate the model so that you can see the bottom of the handle and the bottom of the base. Zoom in so you get a good view of the thickness of the sheetmetal. Make sure the Mate command is still active. Select the two edges as shown in the illustration. The function mate 'Coincident' is selected automatically. Click on OK.	
51	Now, try to drag the handle. You will notice that you can shift it along the edges you have just selected and you can also rotate it around this edge.	
Tip!		Notice that there is a difference between rotating a part of the assembly and rotating the model itself. • To rotate/shift a part you must drag it. You can also use the buttons 'Move Component' and 'Rotate Component'. You can shift a part in relation to the other parts of the assembly. The model changes. • If you rotate the model, the parts remain at the same position in relation to each other, but you will be looking at the model from another angle. The model does NOT change. To do so, you can use the scrollwheel of the mouse (push it and rotate), or you can use the Rotate View command in the View Toolbar.

52	We are going to join the center points of the edges together. Be sure the Mate command is active. Select both center points. When you move the cursor on top of an edge, the center point will appear and you can select it. The mate type 'Coincident' is selected automatically. Click on OK.	
53	Now, try to shift the handle again. Notice that you can only rotate it around the edge but it is fixed in the middle.	
54	We will add the last mate to fix the handle completely. Rotate the model so you have a clear view on both planes as in the illustration and select both of them. The mate type 'Coincident' is selected automatically. Click on OK.	
55	Click on the OK again to close the Mate command.	

56	The candlestick is ready now. Save it as Candlestick.SLDASM.	

	What are the main features you have learned in this tutorial?	In this exercise, you have learned several ways to create parts from sheetmetal.

- You have seen that a 'Base-Flange' is always the first step. In this step you determine the thickness of the material.

- On a 'Base-Flange', you can use the edge flange command.

- With a sketched bend you can create bending lines in the straight plane.

- You have also seen that you can easily make a 2D drawing out of the 3D model by unsuppressing the last feature.

Also you have used some new commands in creating sketches:

- Centerpoint Arc and Tangent Arc to draw parts of a circle.

- Convert to use an existing part in a sketch again.

Finally, you have made a few tricky mates in the assembly.

Slowly you are getting to know SolidWorks better and better, because SheetMetal is an important part of SolidWorks software.

SolidWorks works in education.

One cannot imagine the modern technical world without 3D CAD. Whether your profession is in the mechanical, electrical, or industrial design fields, or in the automotive industry, 3D CAD is THE tool used by designers and engineers today.

SolidWorks is the most widely used 3D CAD design software in Benelux. Thanks to its unique combination of features, its ease-of-use, its wide applicability, and its excellent support. In the software's annual improvements, more and more customer requests are implemented, which leads to an annual increase in functionality, as well as optimization of functions already available in the software.

Education
A great number and wide variety of educational institutions – ranging from technical vocational training schools to universities, including Delft en Twente, among others – have already chosen SolidWorks. Why?

For a **teacher** or **instructor,** SolidWorks provides user-friendly software that pupils and students find easy to learn and use. SolidWorks benefits all training programs, including those designed to solve problems as well as those designed to achieve competence. Tutorials are available for every level of training, beginning with a series of tutorials for technical vocational education that leads students through the software step-by-step. At higher levels involving complex design and engineering, such as double curved planes, more advanced tutorials are available. All tutorials are in English and free to download at www.solidworks.com.

For a **scholar** or a **student**, learning to work with SolidWorks is fun and edifying. By using SolidWorks, design technique becomes more and more visible and tangible, resulting in a more enjoyable and realistic way of working on an assignment. Even better, every scholar or student knows that job opportunities increase with SolidWorks because they have proficiency in the most widely used 3D CAD software in the Benelux on their resume. For example: at www.cadjobs.nl you will find a great number of available jobs and internships that require SolidWorks. These opportunities increase motivation to learn how to use SolidWorks.

To make the use of SolidWorks even easier, a Student Kit is available. If the school uses SolidWorks, every scholar or student can get a **free download** of the Student Kit. It is a complete version of Solid-

Works, which is only allowed to be used for educational purposes. The data you need to download the Student Kit is available through your teacher or instructor.

The choice to work with SolidWorks is an important issue for **ICT departments** because they can postpone new hardware installation due to the fact that SolidWorks carries relatively low hardware demands. The installation and management of SolidWorks on a network is very simple, particularly with a network licenses. And if a problem does arise, access to a qualified helpdesk will help you to get back on the right track.

Certification
When you have sufficiently learned SolidWorks, you can obtain certification by taking the Certified SolidWorks Associate (CSWA) exam. By passing this test, you will receive a certificate that attests to your proficiency with SolidWorks. This can be very useful when applying for a job or internship. After completing this series of tutorials for VMBO and MBO, you will know enough to take the CSWA exam.

Finally
SolidWorks has committed itself to serving the needs of educational institutions and schools both now and in the future. By supporting teachers, making tutorials available, updating the software annually to the latest commercial version, and by supplying the Student Kit, SolidWorks continues its commitment to serve the educational community. The choice of SolidWorks is an investment in the future of education and ensures ongoing support and a strong foundation for scholars and students who want to have the best opportunities after their technical training.

Contact
If you still have questions about SolidWorks, please contact your local reseller.

You will find more information about SolidWorks at our website: http://www.solidworks.com

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