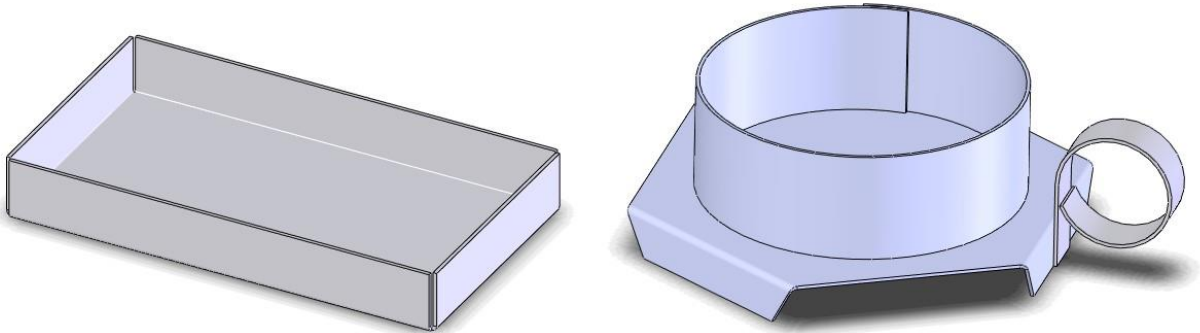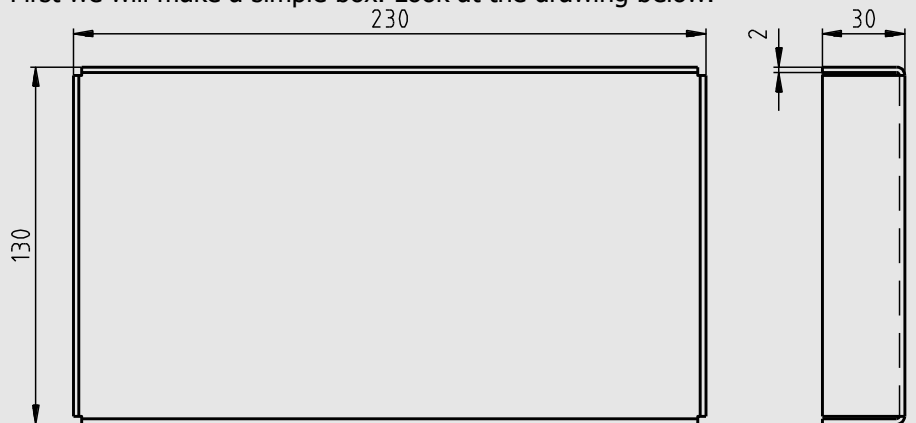# SOLIDWORKS® tutorial 4

CANDLESTICK

In this tutorial you will make a simple container and after that a candlestick out of sheet metal. You will get to know working with sheet metal in SOLIDWORKS. We will show you a couple of ways to create a product out of sheet metal and we will show you how to make a drawing in 2D.

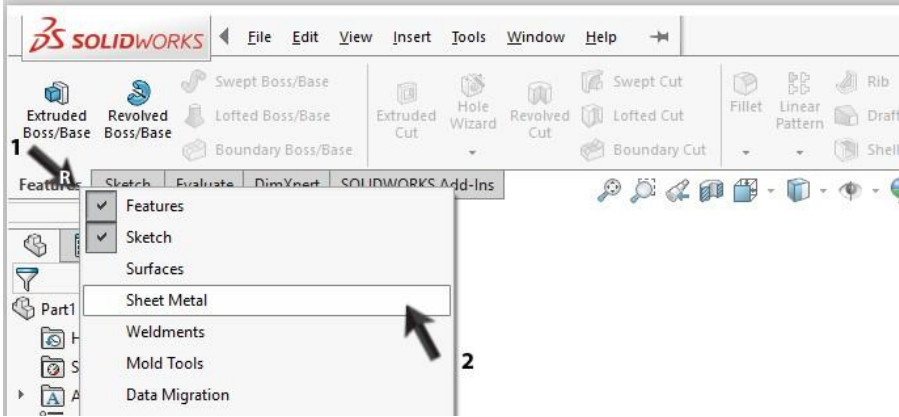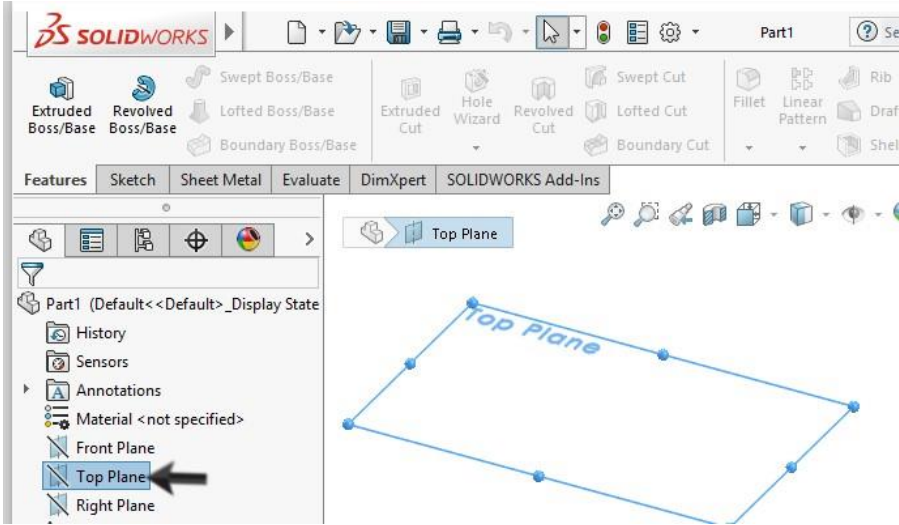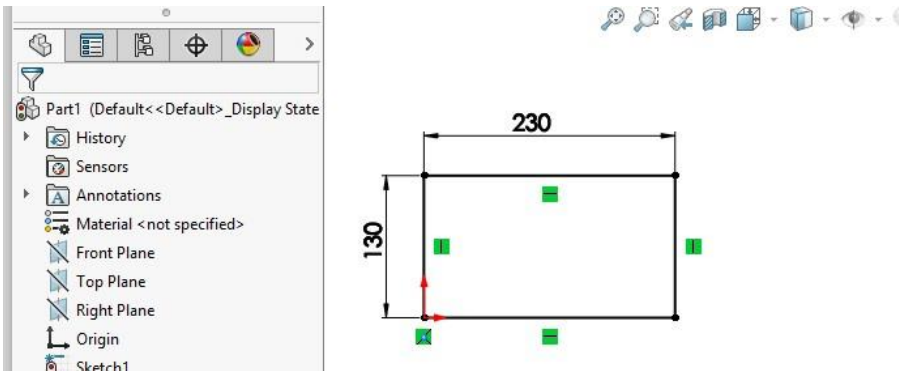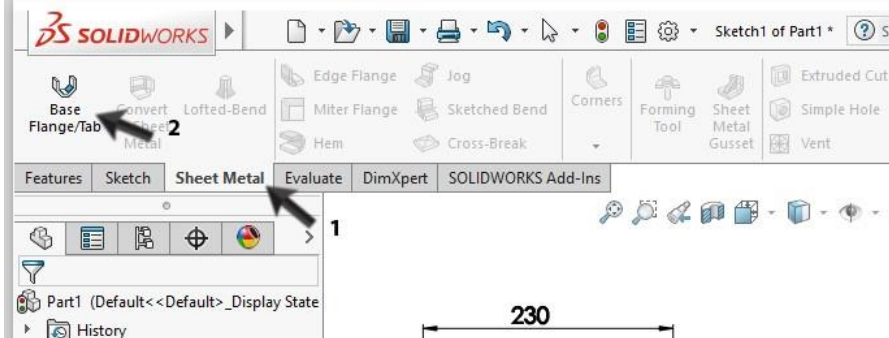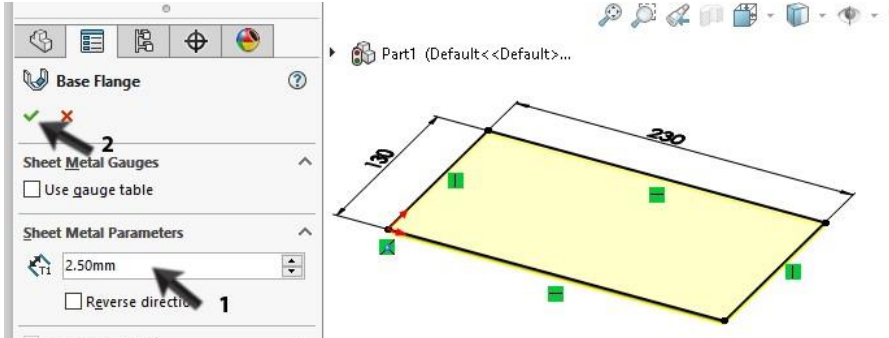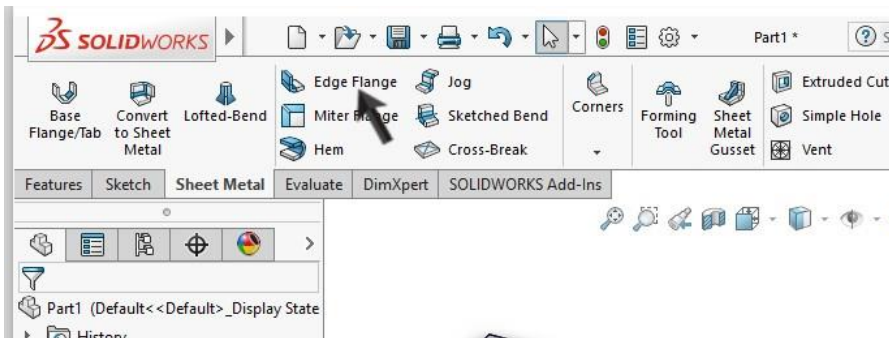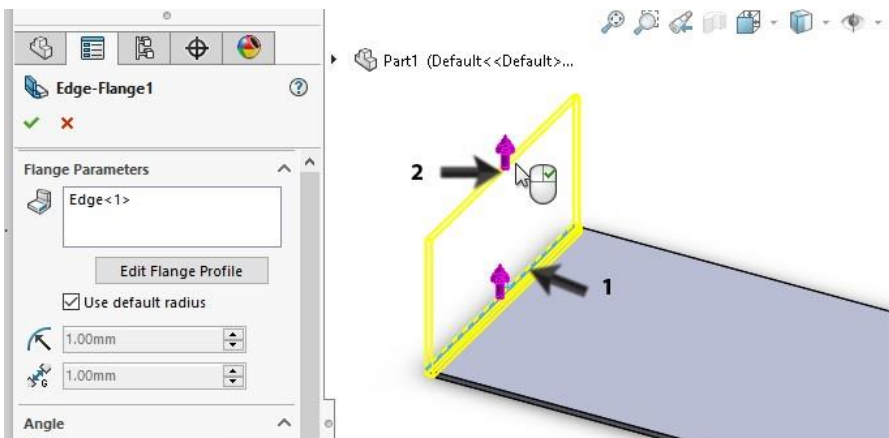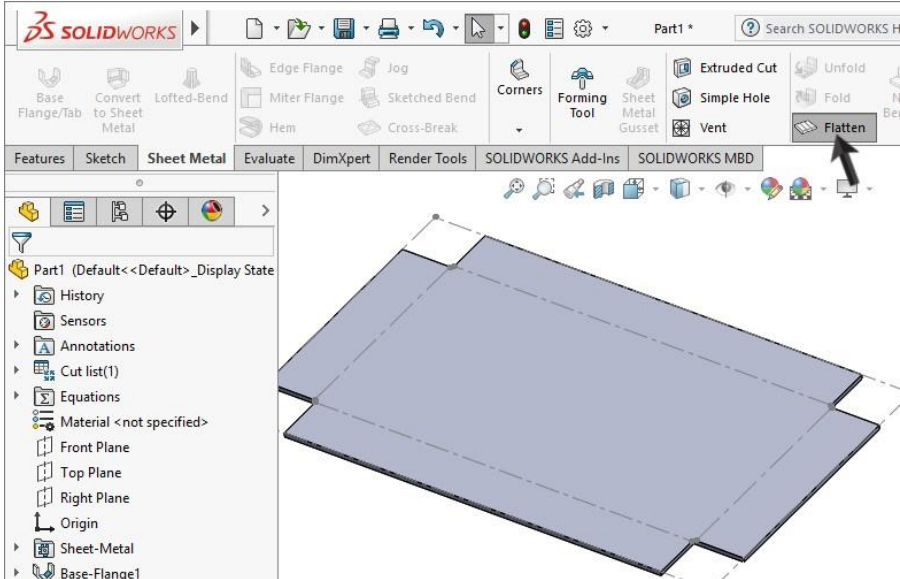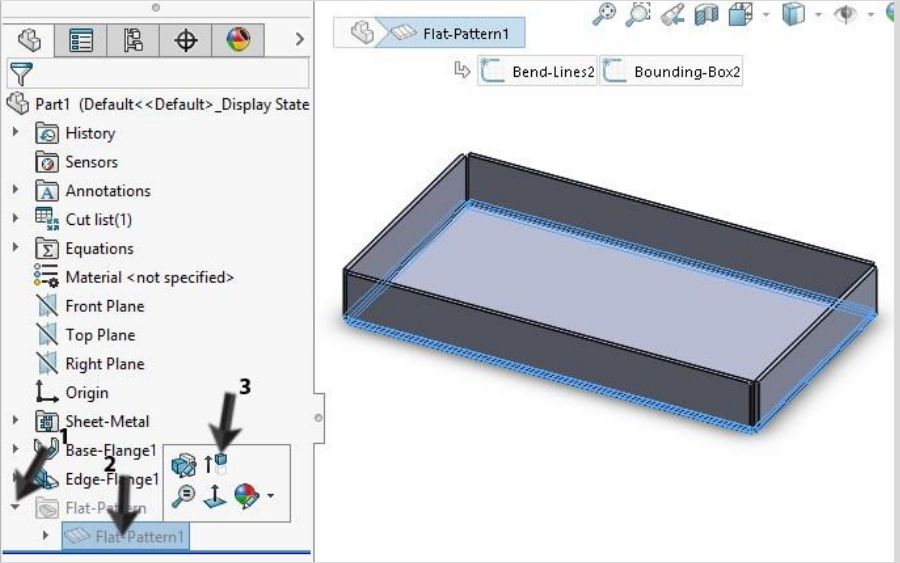| **Werkplan**	First we will make a simple box. Look at the drawing below.
	230 2 30
	130
	We will execute following steps:
	1. First we will create the base, for this we will use an outside dimension of 230 x 130
	2. After that we will add four sides with a height of 30.
	3. Finally we will create a flat pattern and a drawing.

1	Start SOLIDWORKS and open a new part.	

2	Make sure the buttons you need to work with Sheet Metal are visible. The easiest way to get these working 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 corer at the origin. Add dimensions and set the height to 130 and width to 230. Do you still remember how to start a sketch? If not, look at step 2 and 3 of tutorial #3.	

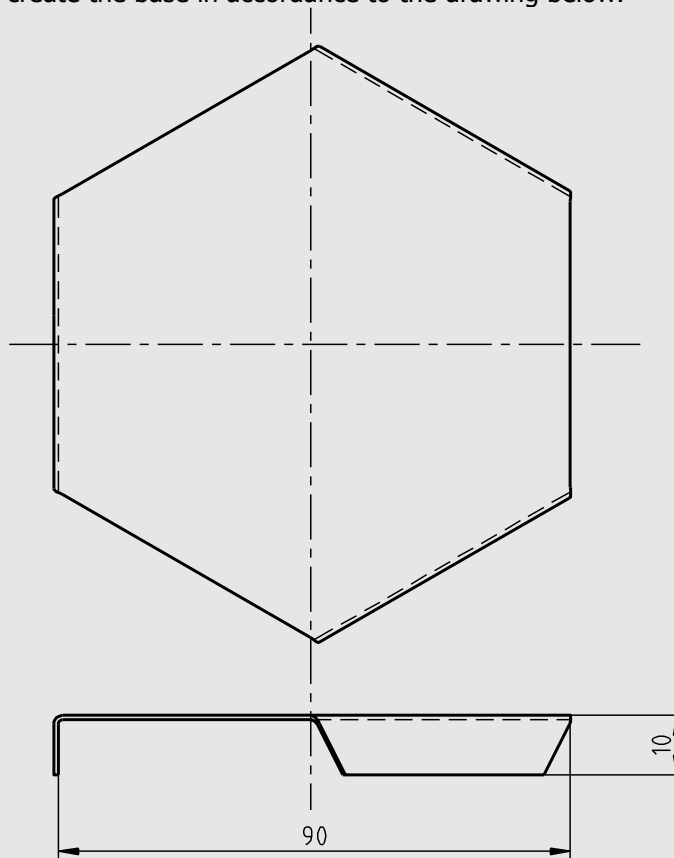|		
5	Next click on the tab Sheet Metal in the CommandManager next on Base-Flange	
6	1. Set the thickness at 2 mm in the PropertyManager. 2. Click OK.	
7	To create the edges of the container, click on Edge Flange in the CommandManager	
8	1 Click on the first edge of the base and move the mouse upwards. 2 Set the first wall 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 like it is shown in the illustration at the 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 OK.	
10	The box is ready now. Next we will take a look at the flat pattern (the shape that has to be cut out of sheetmetal to create this box). Click on the button Flatten in the CommandManager.	

11	At this point, a 2D draw of the container is visible If you want to return to t normal view in 3D, cl Flatten again.	
	Tip!	
		When clicking the button Flatten, the only thing that happens is that the last feature in the FeatureManager is being suppressed. This feature (FlatPattern) is created automatically in a sheet metal part, and cannot be deleted. If you wish, you could also (un)suppress this feature directly to show/hide the flat pattern.
12	Save the model as: box.SLDPRT	

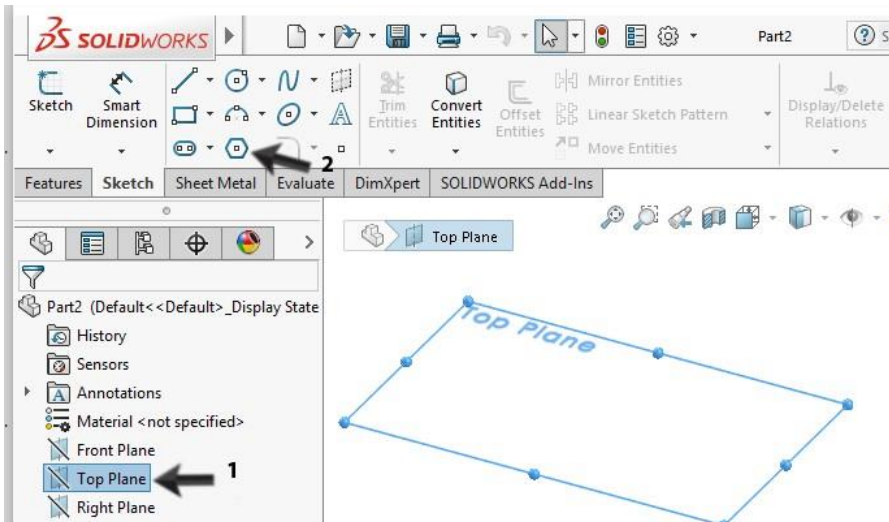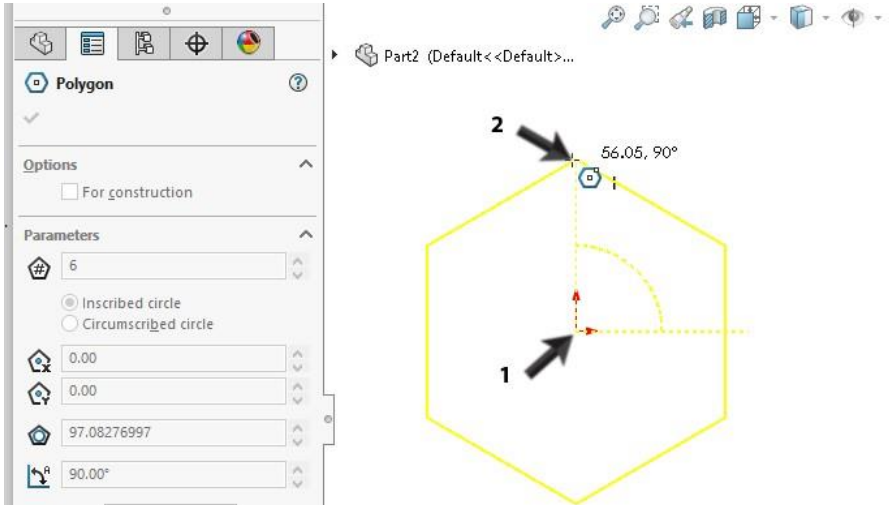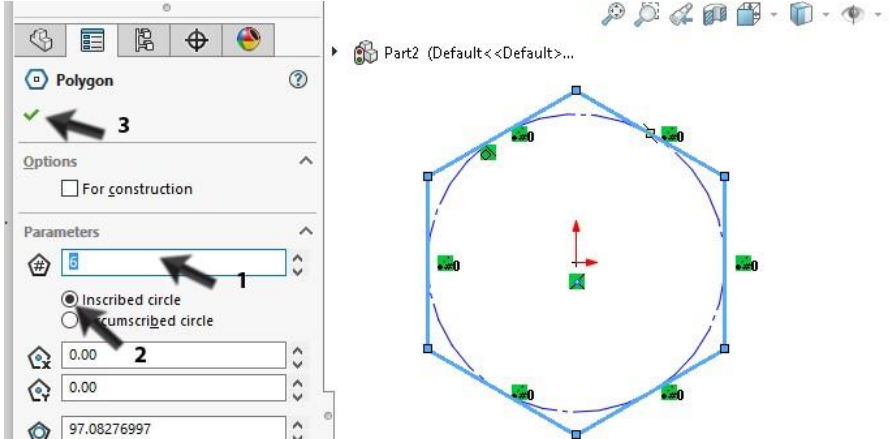|		
	Work plan	We are going to create a candle stick. It consists of three parts. First, we will create the base in accordance to the drawing below.
		90 10
		The handling of this product is different from the other ones. We draw a 2D drawing and bring in some bending lines. The hardest part of this model is to make the first sketch.
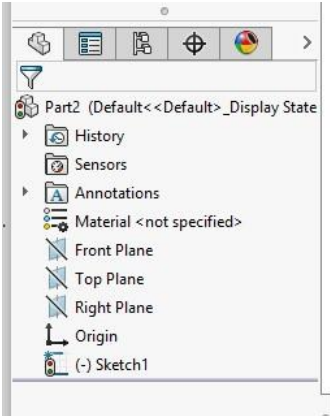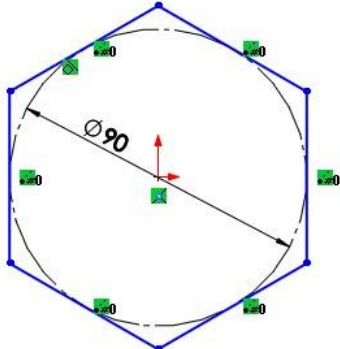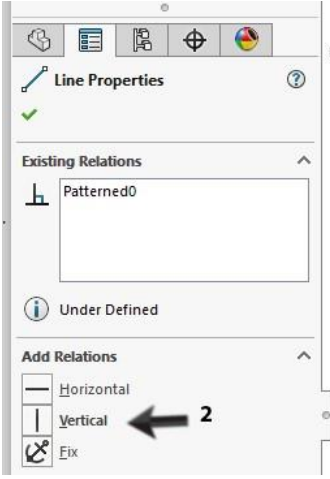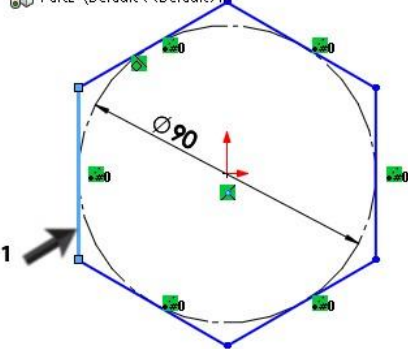
13	Open a new part	

14	1. Select the Top plane, to make a sketch on it. 2. Click on Polygon in the CommandManager.	
15	Click on origin for the center point of the hexagon and at a point straight above the origin at a random distance from the first corner.	
16	Be sure that in the PropertyManager: 1. The number of sides is set to 6 2. The inscribed circle is selected 3. Click OK.	

17	Set the dimension of the inner circle to 90 mm with Smart Dimensions.	
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.	

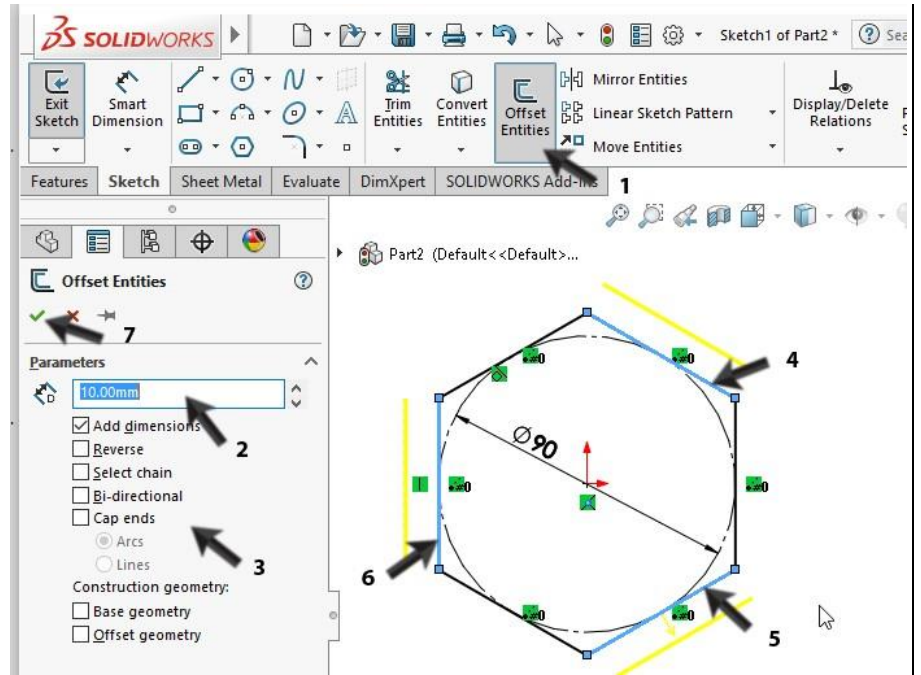|		
	3 Copy the other settings of the PropertyManager from the drawing at the right. Be sure the option Select Chain is NOT selected. 4-6 Select the sides of the hexagon as shown at the right. Note: when the lines are off-setted to the inside, check the option 'Reverse' in the PropertyManager. 7. Click OK.	
20	1. Click on Trim Entities in the CommandMan-ager 2. Select the option Corner in the PropertyManager 3-4 Click in the sketch on two lines which have to form a corner.	
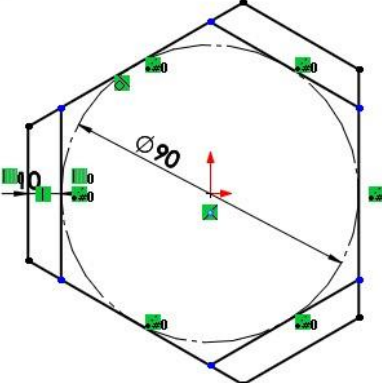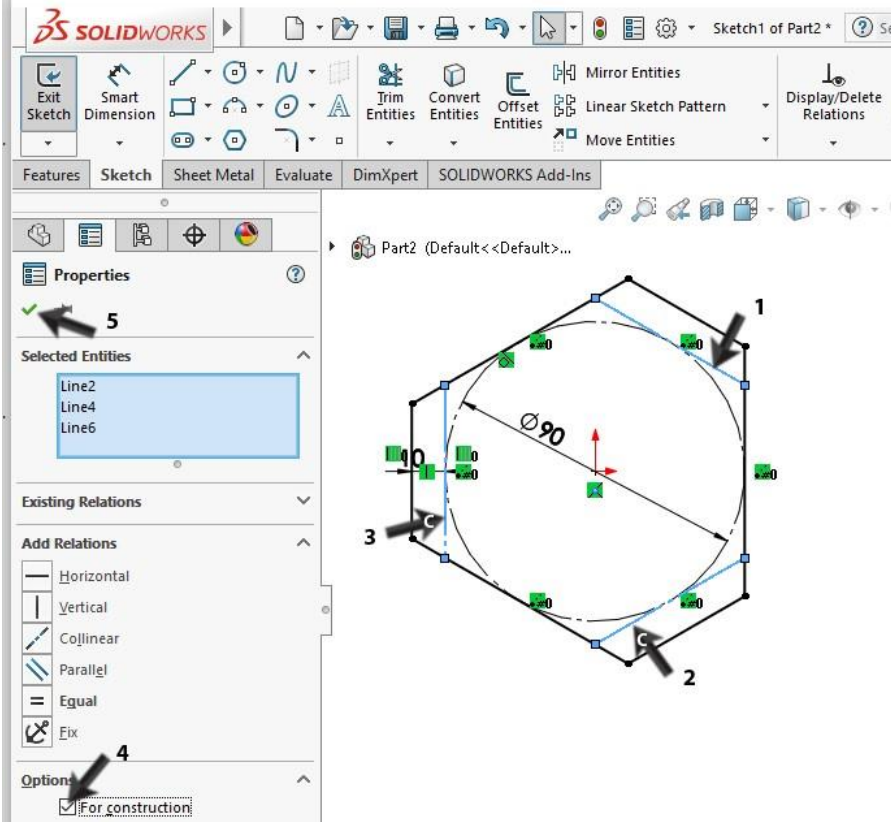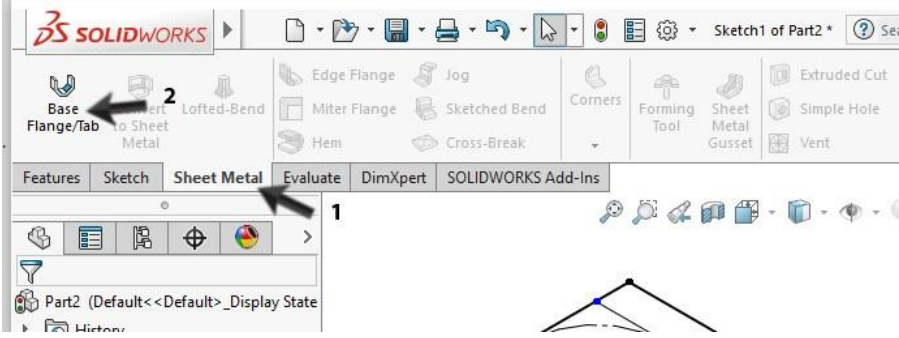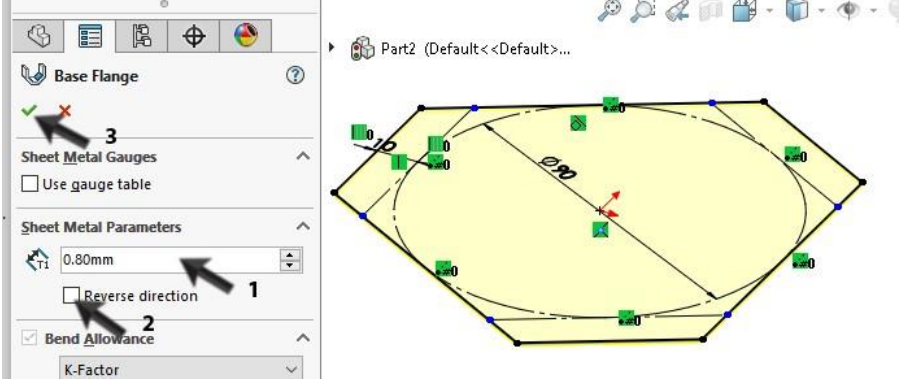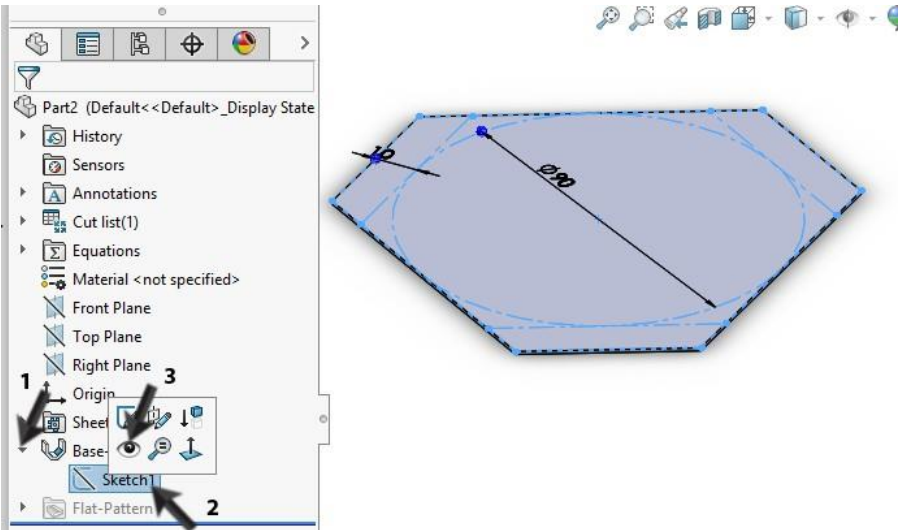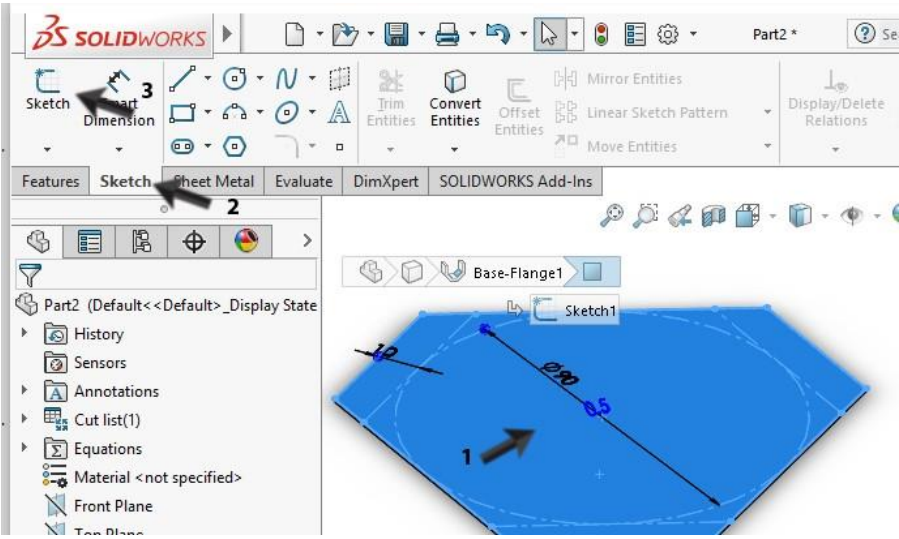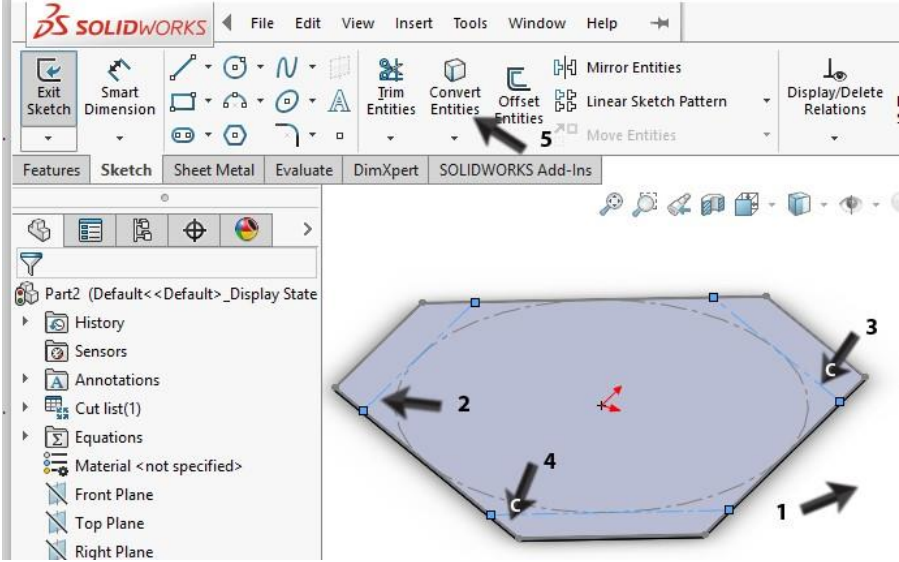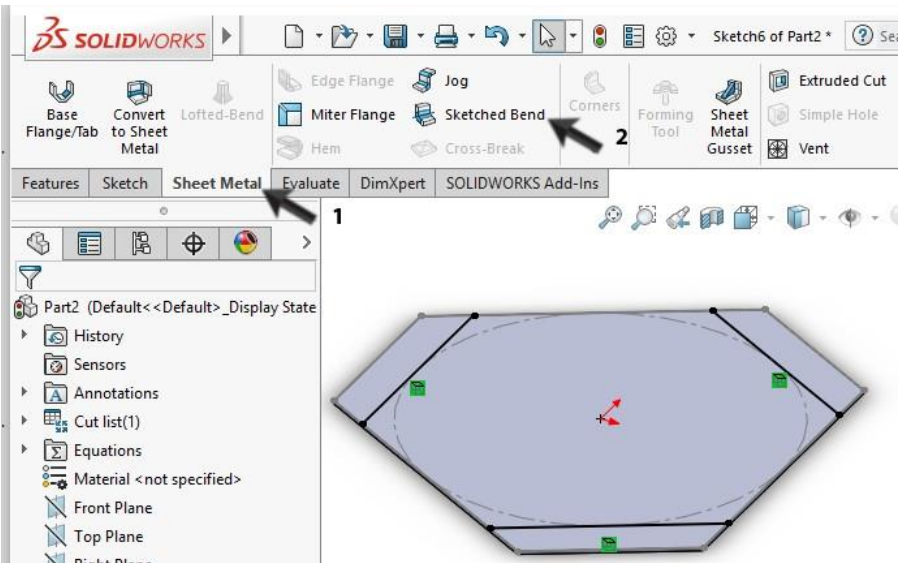
21	Click two lines again and again so you see the drawing as shown at the right.	
22	Finally we will transform the three inner lines to construction lines. This will be the bending lines later on. 1-3 Select the three lines (use the <ctrl>-button on your keyboard) 4. Check the option For construction in the PropertyManager. 5. Click OK.	
23	t create the base. Click on Sheet Metal in the CommandManager. 1.	

	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 sketch. Have you got a good view at the material? If not, zoom in! 3. Click 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 Click on the sketch which is visible now. (usually this is: Sketch1) 3 Click on Show in the menu that appears. The sketch is now grey colored in the model.	

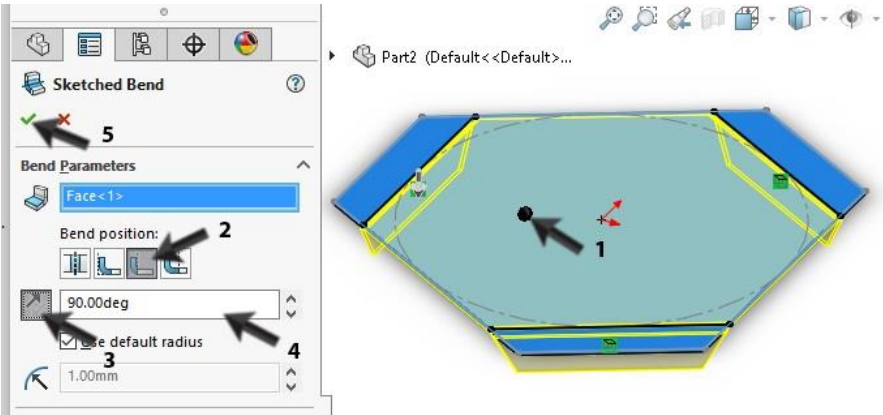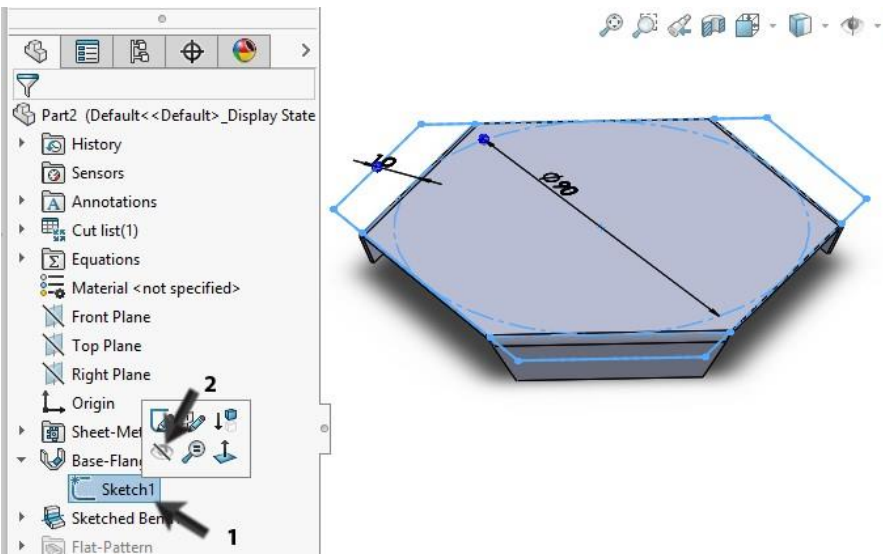| 26	Start a new sketch at the top plane:	
	1. Select the top plane of the item you have just created	
	2. Click on Sketch in the CommandManager to show the right buttons.	
	3. Click on the Sketch command to open the sketch.	

	Tip!	In earlier exercises we opened a sketch selecting a plane and draw a rectangle (example). SOLIDWORKS 'understands' in such a case that you want to open a sketch and does so automatically. Before you can use the command from the next step, a sketch **must be** open already, or else 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 besides the model to deselect 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	

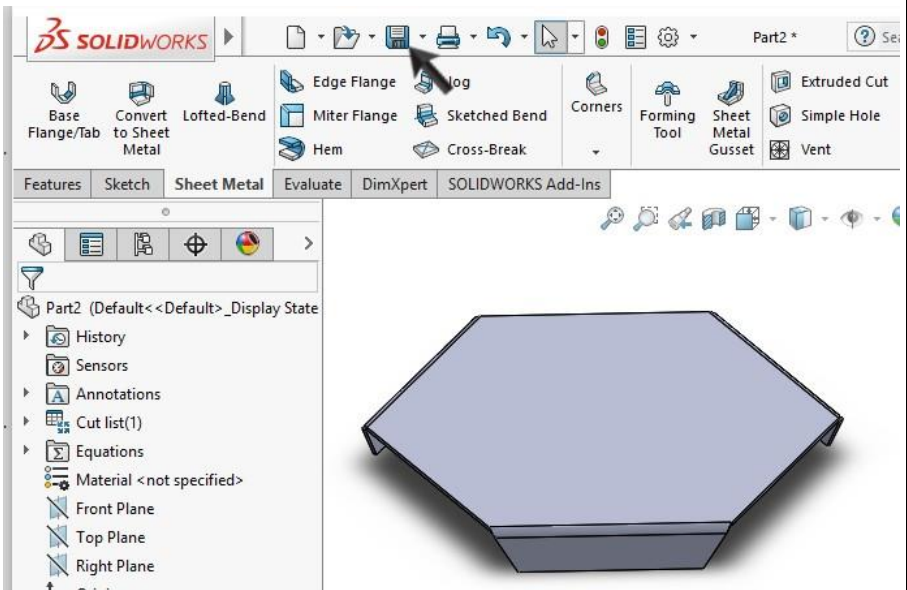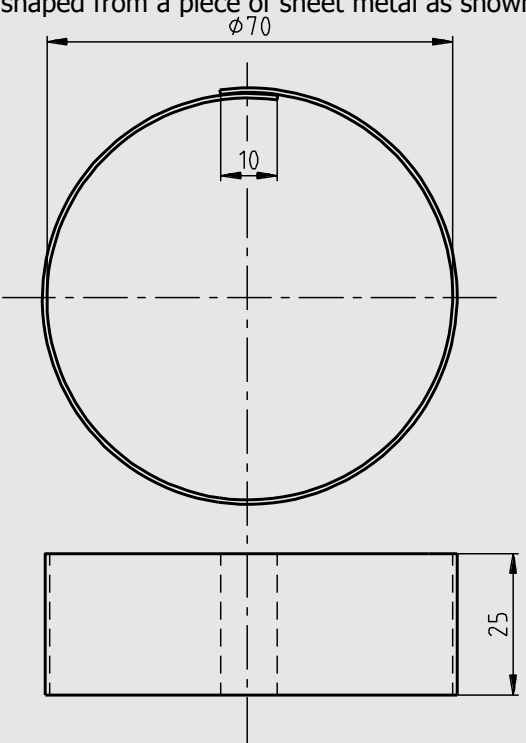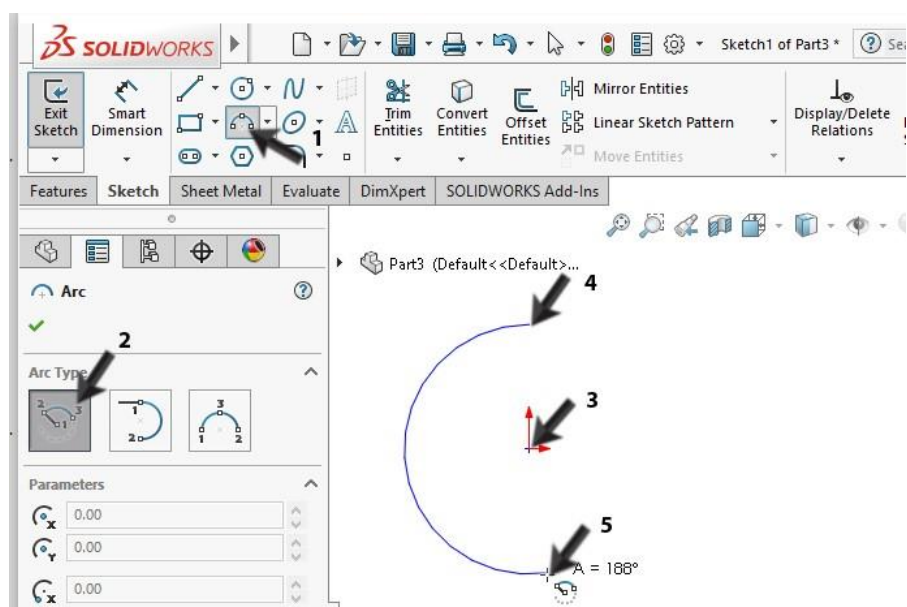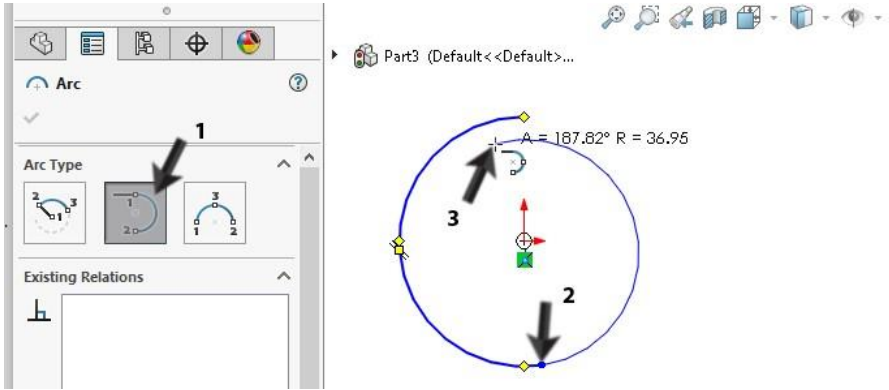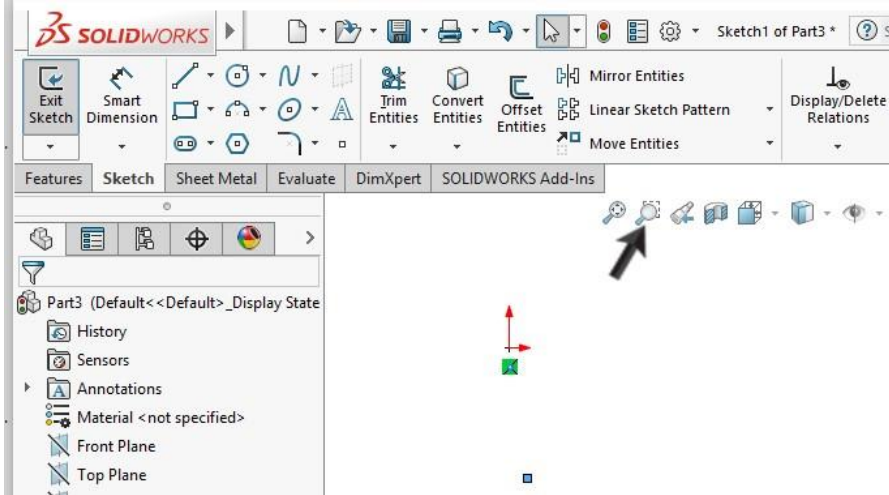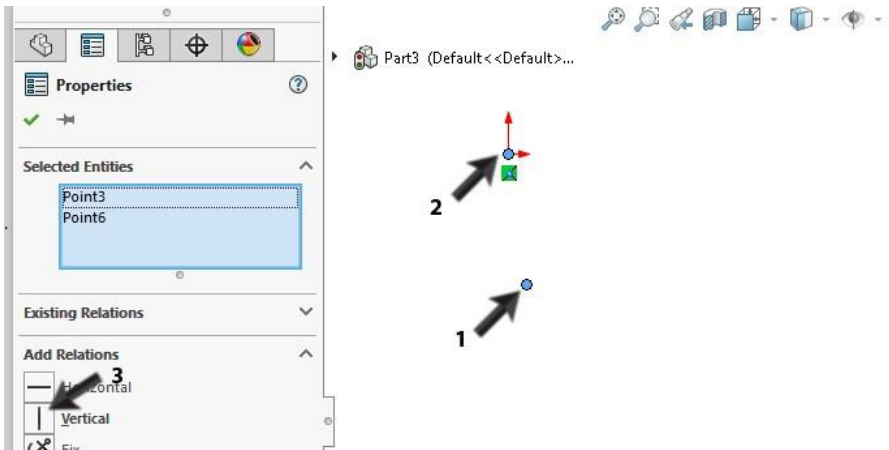|	Tip!	For most features in SOLIDWORKS you must first make a sketch. So you can't use an edge or an existing line to use them in a new feature.
		But you CAN do what we have just done here: to make a copy of existing element and paste them in a new sketch. This can be a line from an old sketch but it can also be an edge of a model or even a face. In this way it is a very fast way to make a new sketch which is derived from the existing model.
		When an element is not exactly in the plane of the sketch, it will be projected on it.
28	1. Click on Sheet Metal in the CommandManager, 2. Click on Sketched Bend.	
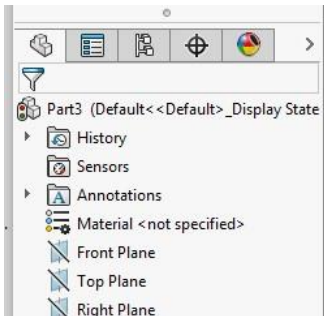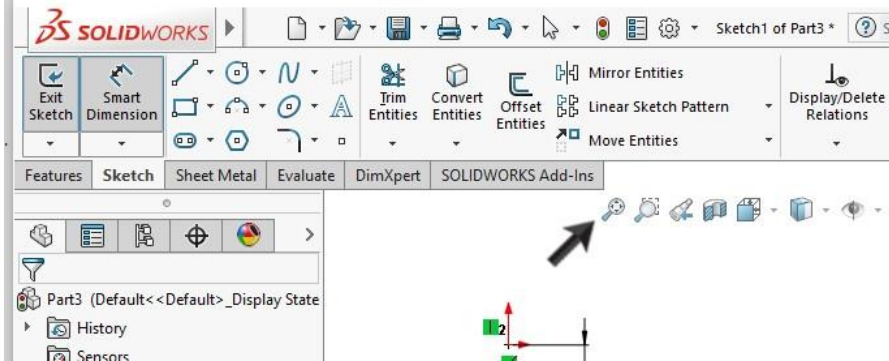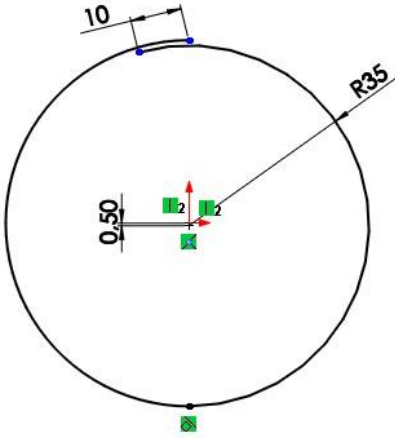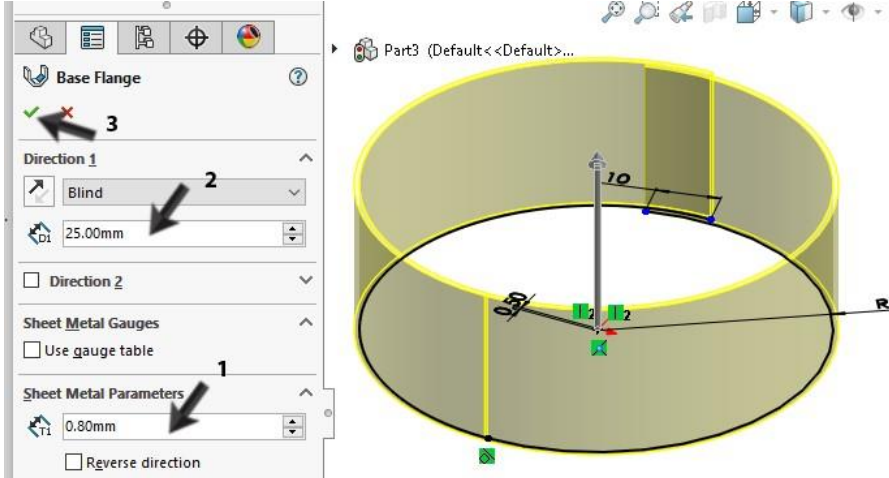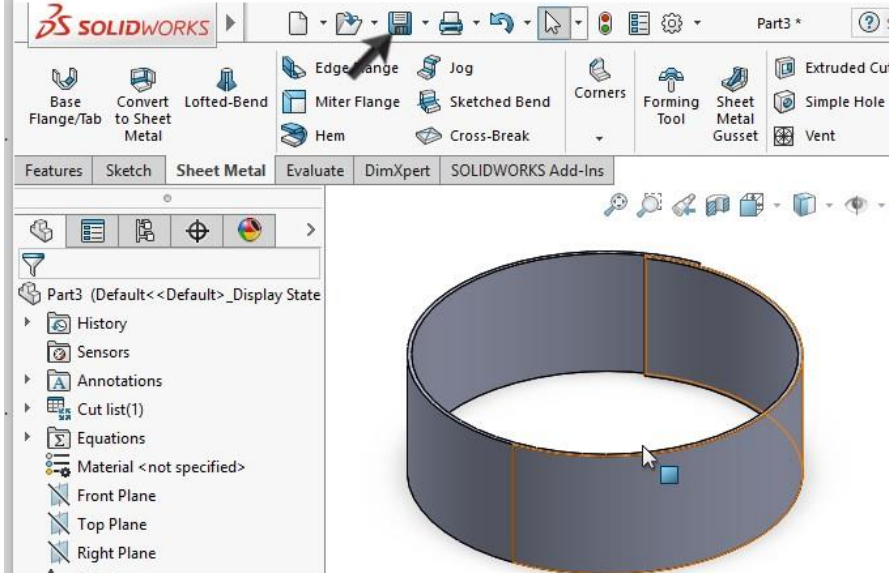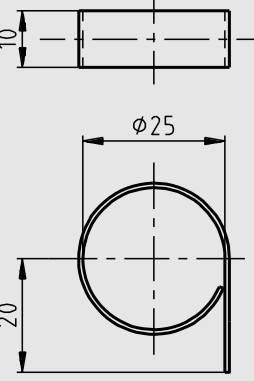
29	1	Click at a position in the middle of the base. With this, you set this part of the base to be fixed. The other parts will bend later on.
	2	Select the option Material outside: this is re- lated to the way in which the dimensions are in the drawing.
	3	With the Reverse direction–button you determine in which direction the material is bent. (up or down) 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 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 candle stick is the 'tube' to put the candle in. This is shaped from a piece of sheet metal 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 Arc in the CommandManager. 2. Select Centerpoint Arc 3. Click on the origin for the first point. 4. Click straight above the origin for the second point. 5. To finish this half, click on a third point, about straight below the origin		
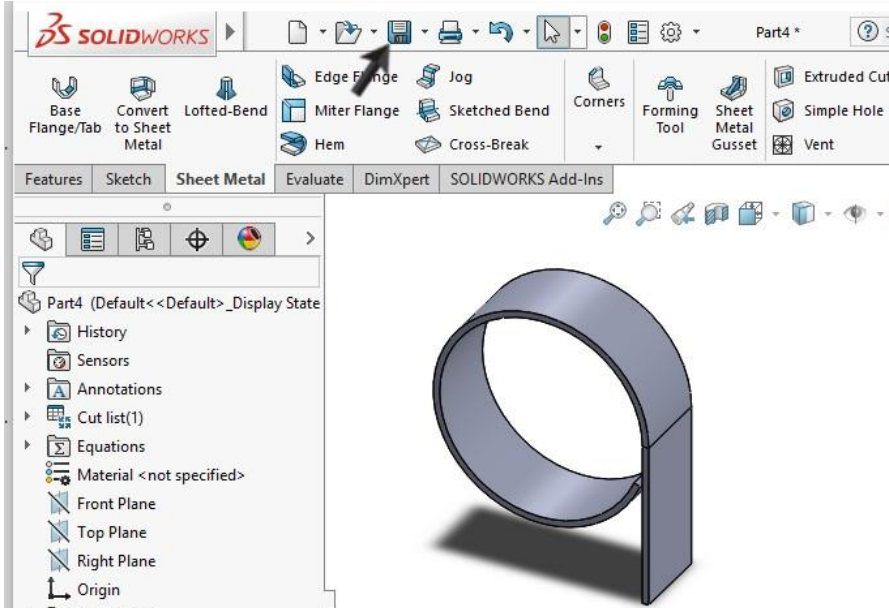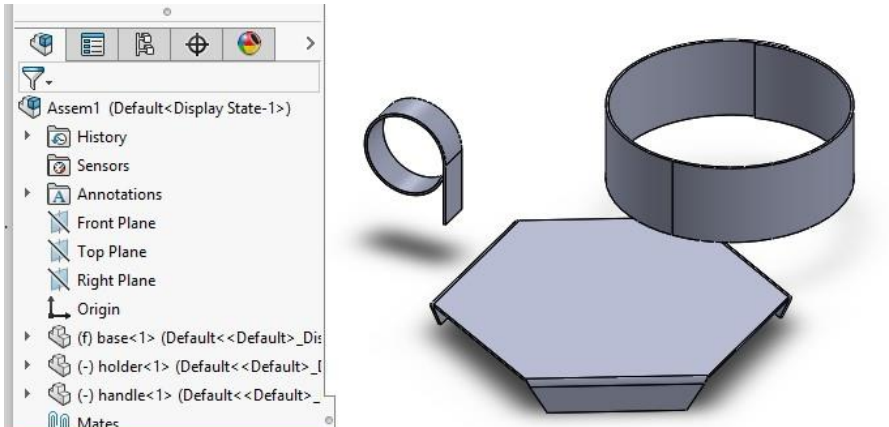
34	Next we will draw the second part of the 'circle'. 1. Click on Tangent Arc in the PropertyManager. 2. Click on the bottom point of the arc just drew first. 3. Click on a point as shown in the illustration. 4. Stop the command by pushing the <esc> button.	
35	Zoom in on the origin of the circle with the centre 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 add a dimension of 0.5mm between both points.	
38	Click Zoom to fit in the View-toolbar to show the entire sketch.	
39	Add two more dimensions tot the sketch with the Smart Dimensions command: 1. A radius of 35 for the right arc. 2. A length of 10mm the first circle to overlap 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 Sheet Metal in the CommandManager and next on Base-flange. Set the following features in the PropertyManager: 1. Thickness of the material 0.8mm 2. Height 25mm 3. Click OK	
41	The cylinder is ready now. Save the file as hold-er.sldprt	
	Work plan	Finally we have to make the handle of the candle stick. This is done in exactly the same way as we did for the last part. It is again the most important to make a sketch.
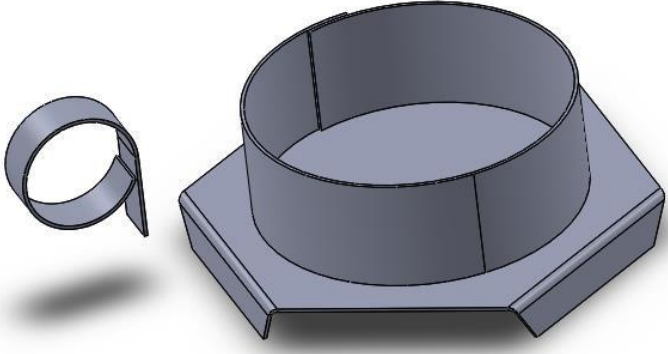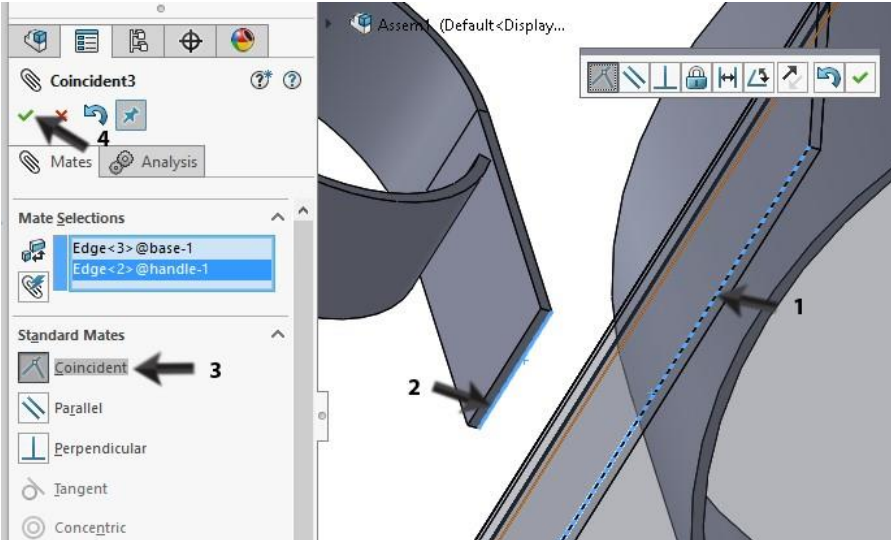
42	Open a new part and start drawing a sketch at the Front Plane. Draw a line from the origin up. Use the Tangent Arc command to draw a part of a circle (an arc) like it is shown in the illustration.	
43	Add three dimensions with Smart Dimension like in the illustration on the right.	
44	Use the Base-flange command to set the thickness of the material to 0.8mm and a height of 10mm.	
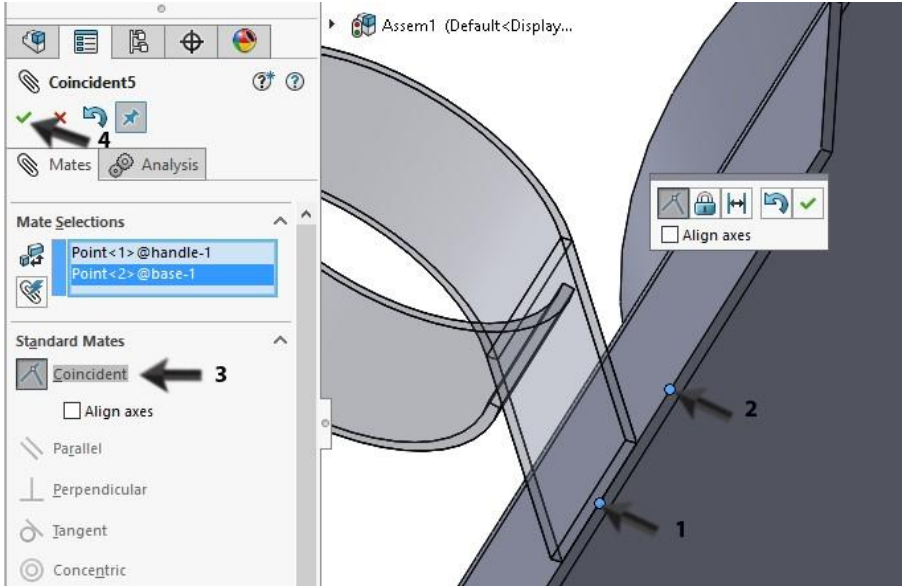
45	Save the file as han-dle.sldprt	
		To complete this tutorial we will make an assembly. We have done this before. Would you be able to join the three parts together in an assembly? Try it yourself first, before you continue reading!
46	Open a new assembly. Use the Insert Component to place the base 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 step 46 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 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.
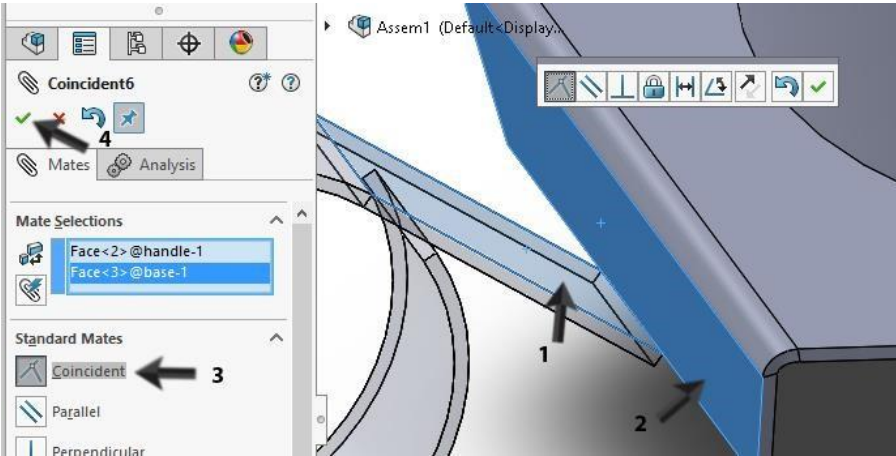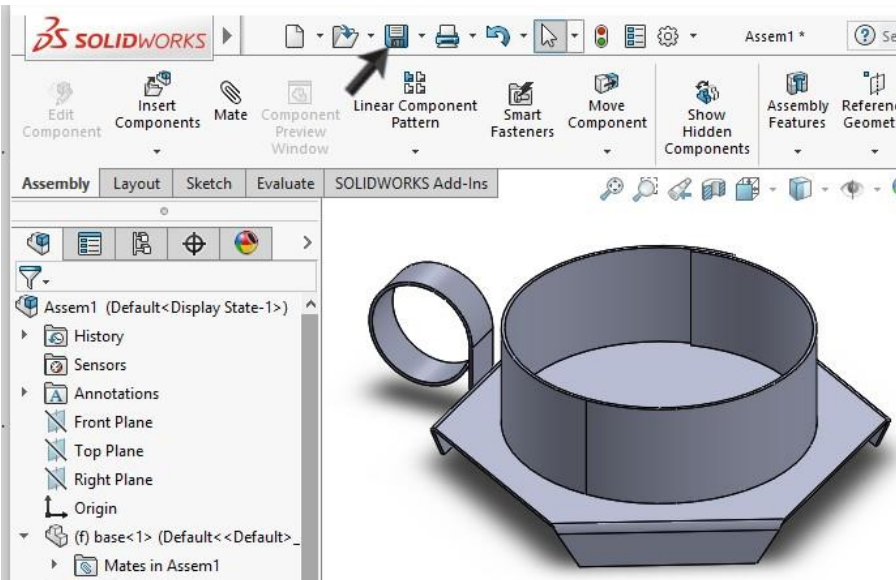
		When you click on OK twice, the Mate-command will end. We assume that we stay within the Mate-command. When you click twice on OK by accident, click on the Mate-command in the CommandManager to start a new Mate.
48	Be sure the Mate-command is active (read the tip above). 1. Select the origin of the base in the Feature Tree 2. Also select the origin of the holder. 3. The mate type Coincident is again selected automatically. 4. Click OK.	

49	Make sure the handle is placed in the area where it has to be at the end. Look at the illustration at the right. When this part is placed somewhere else, you can drag it to its right position.	
	Tip!	We are using illustrations of the model in which the model is rotated in such a way, that either edges or points which are needed to create a mate, are visible at the same time. This is the most convenient, because there will be no need to rotate the model during the 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 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 sheet metal. Make sure the Matecommand is still active. Select the two edges like it is shown in the illustration. The function mate coincident is selected automatically. Click OK.	

51	Now try to drag the handle: you will notice that you can shift it along the edges we have just selected and it can also rotate 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 will shift a part in relation to the other parts of the assembly. The model changes. Move Component Rotate Component • 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 use the Rotate View command in the View-toolbar.

52	We are going to join the centre points of the edges together. Be sure the Matecommand is active. Select both centre points. When you move the cursor on top of an edge, the centre point will appear and you can select it. The mate type Coincident is selected automatically. Click 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	The last mate we will add to fix the handle completely. Rotate the model so you have a clear view on both planes like in the illustration and select both of them. The mate type Coincident is selected automatically. Click OK.	
55	Click on the OK again to close the Mate-command	
56	The candle stick is ready now. Save it as Candlestick.sldasm.	

Which are the main features you have learned in this tutorial?	In this exercise you have seen several ways to create parts from sheet metal.
	• You have seen that a Base-flange always is the first part. In this you determine – amongst others - 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 2 D drawing out of the 3 D 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 Sheet Metal is an important part of the SOLIDWORKS software.