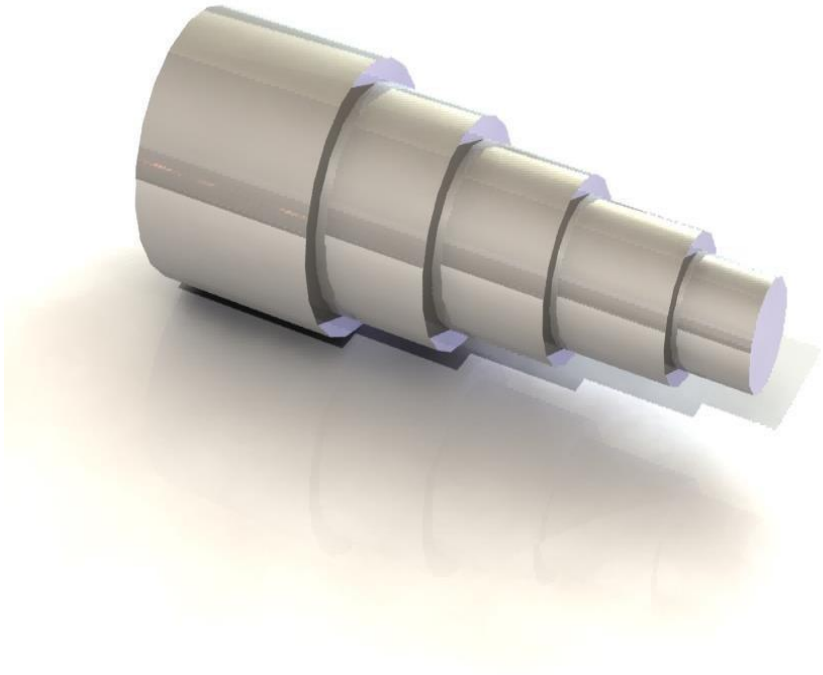
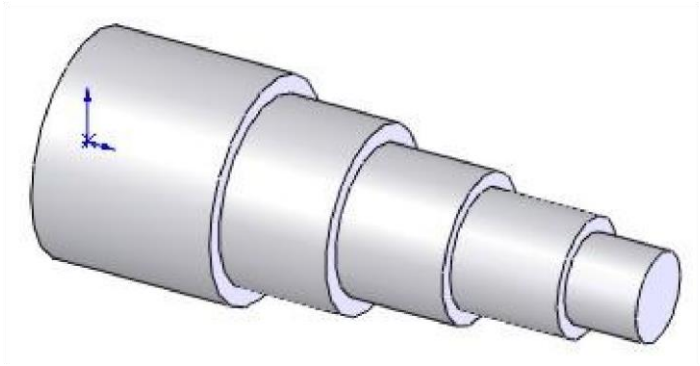
SOLIDWORKS tutorial 1

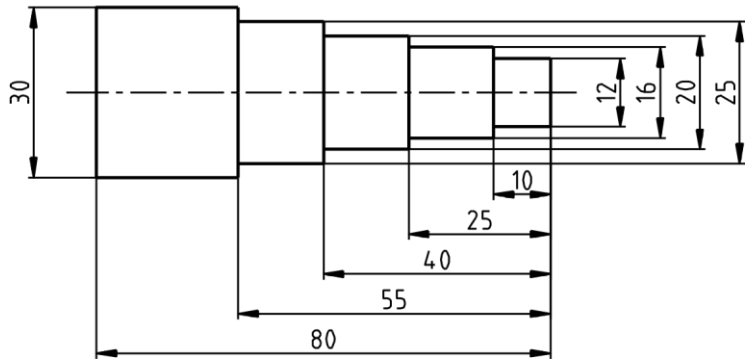
Axis



Axis

This first exercise is meant as an introduction to SOLIDWORKS. First we will design and draw a simple product: an axis with different diameters. You'll learn how to work with the software and you will learn a few basic principles of the software. You will find out how to add material and how to remove it afterwards.





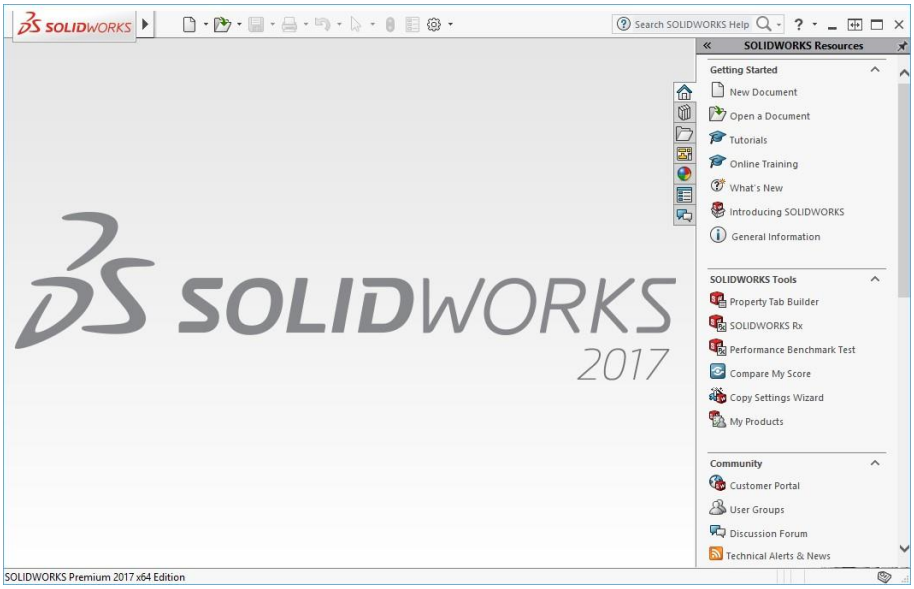
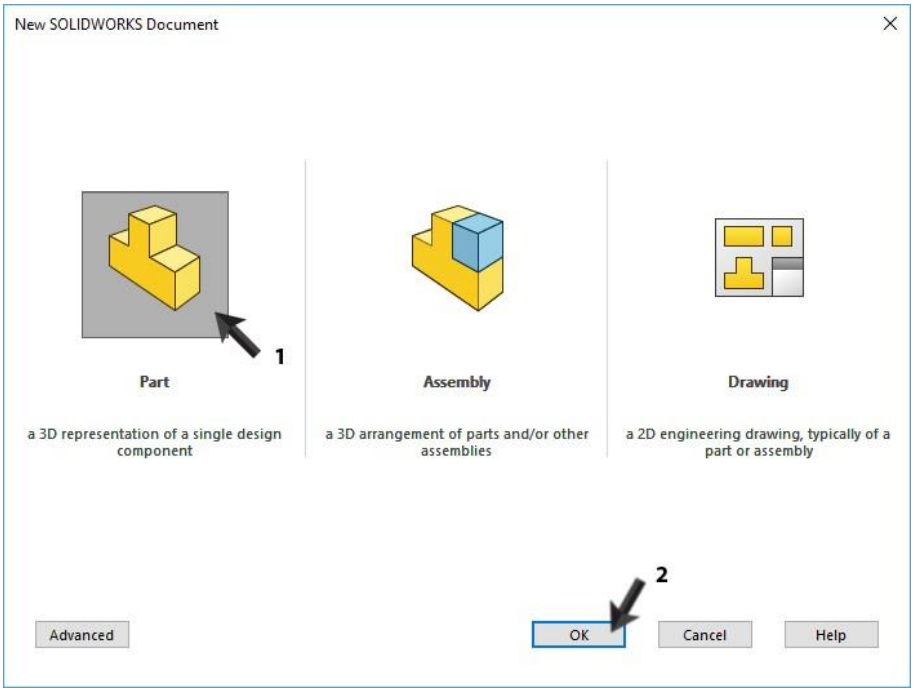
Work plan

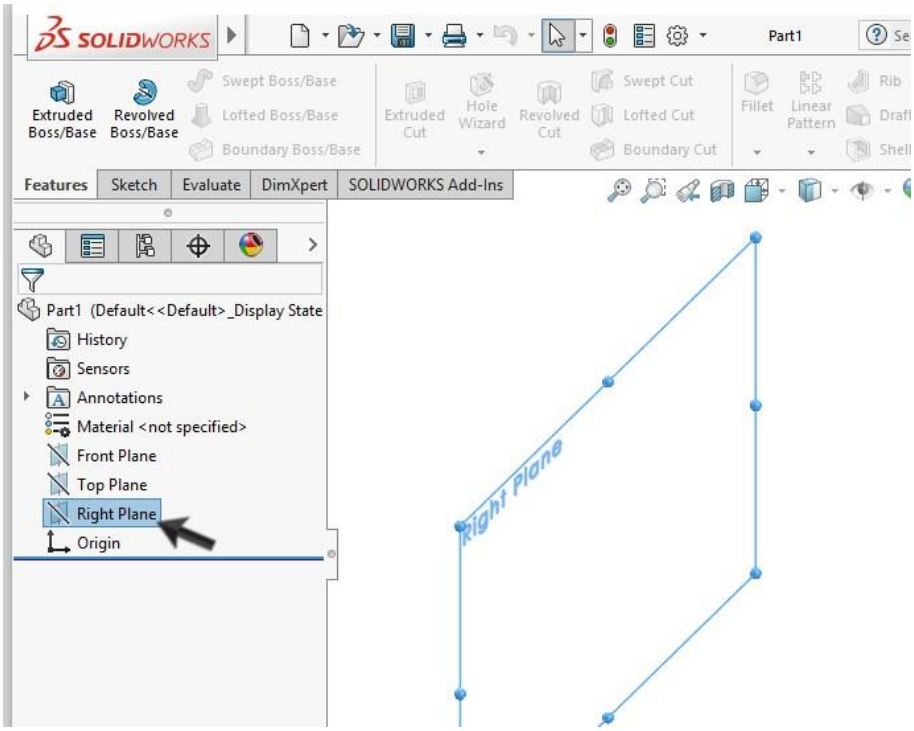
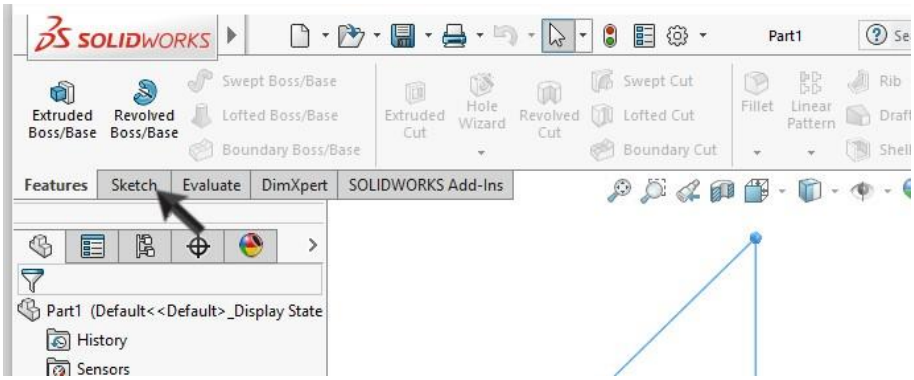
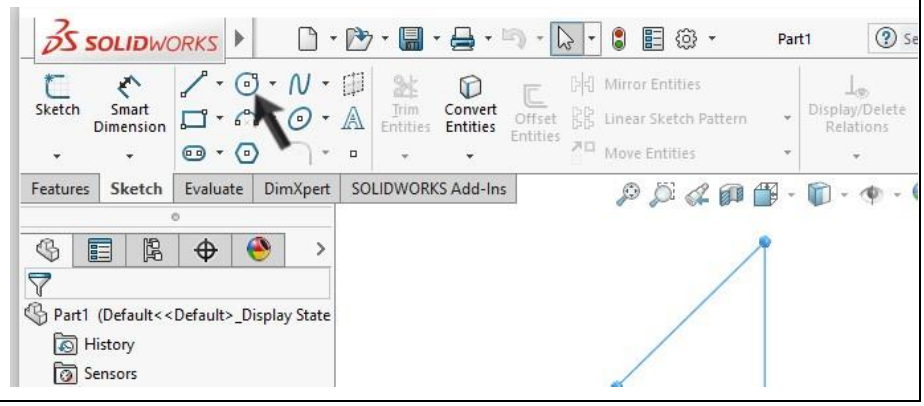
Before you start drawing in SOLIDWORKS, you must have a work plan: how are you going to do it.

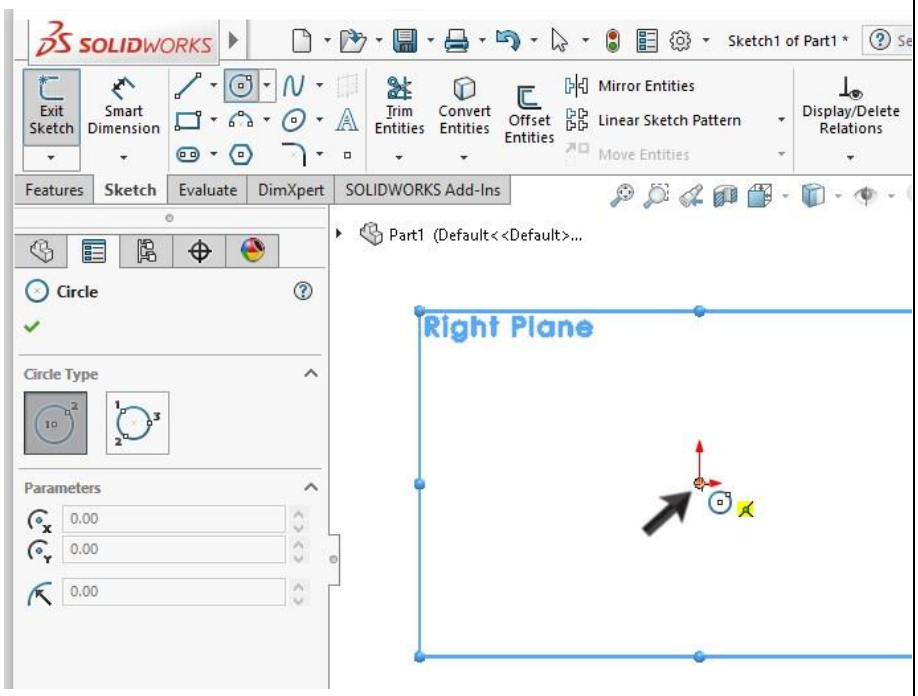
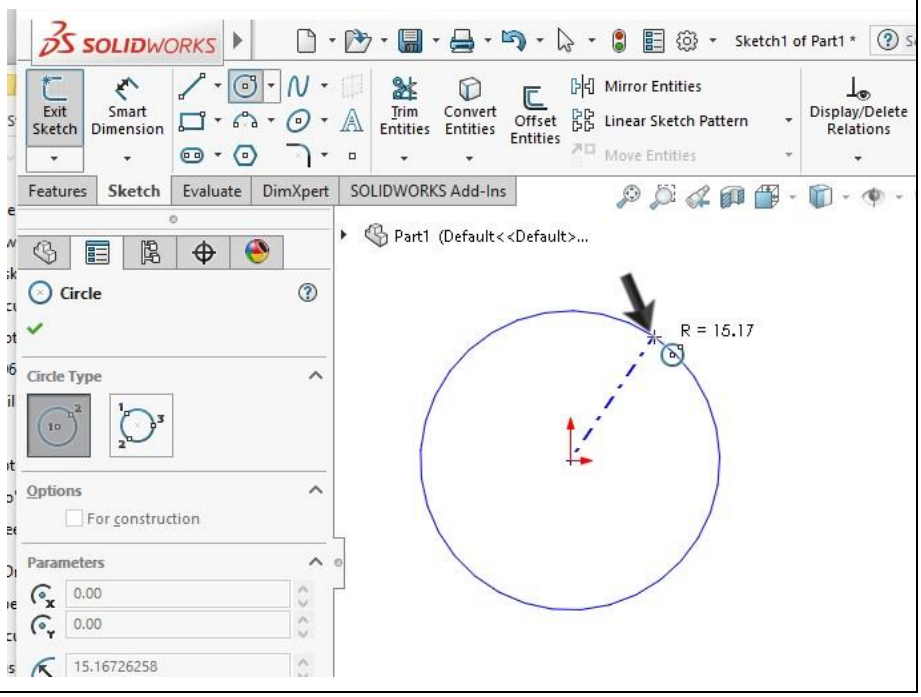
Most time you will produce a part in SOLIDWORKS in the same way as you would do it in the workshop. For this assignment this means you have to go through the following steps:

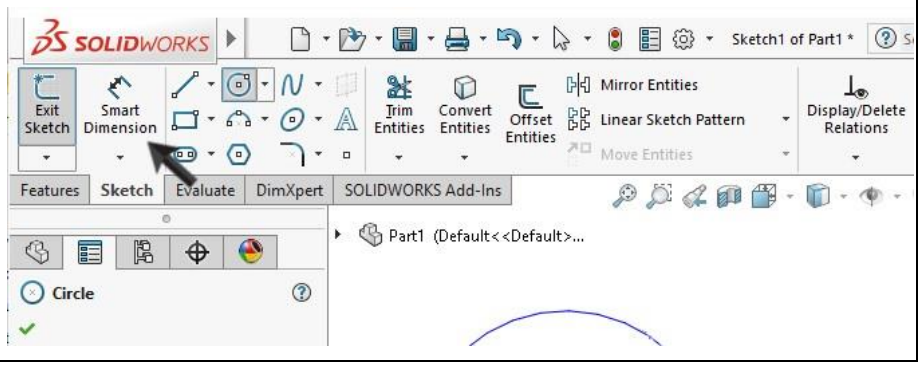
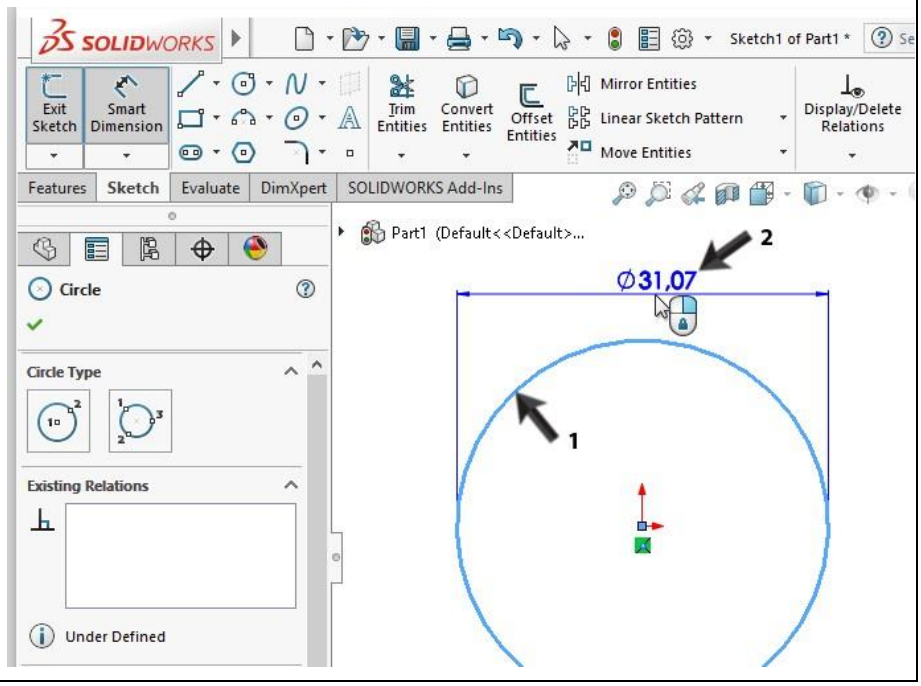
1. Create an axis of Ø30 x 80,
2. Cut the material in order to get the different diameters.

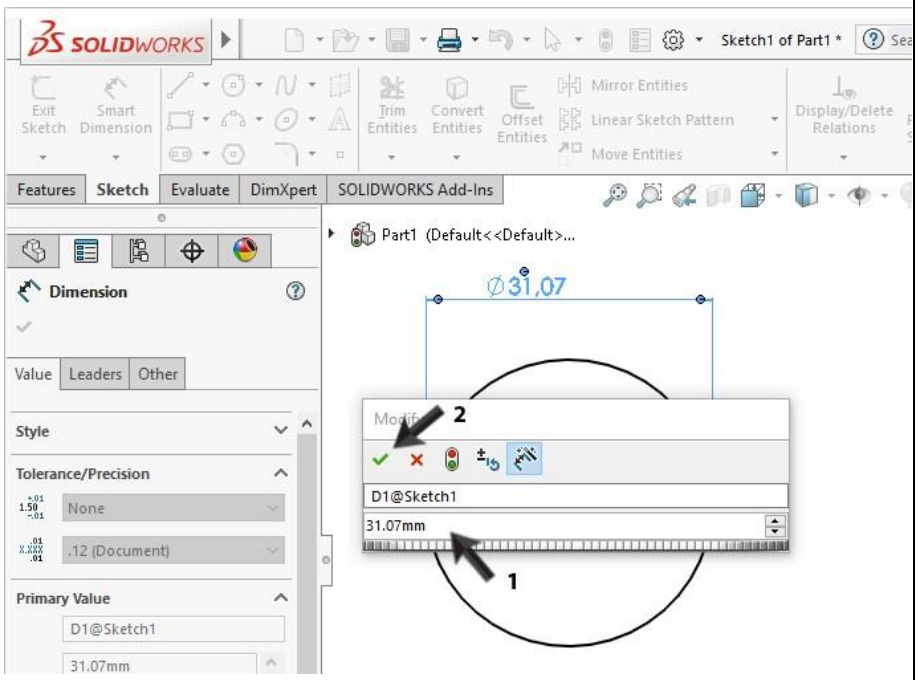
At the lathe you would have to perform several extra steps to get the desired accuracy, because for instance, you cannot remove all the material in one run. In SOLIDWORKS this is no problem, of course.

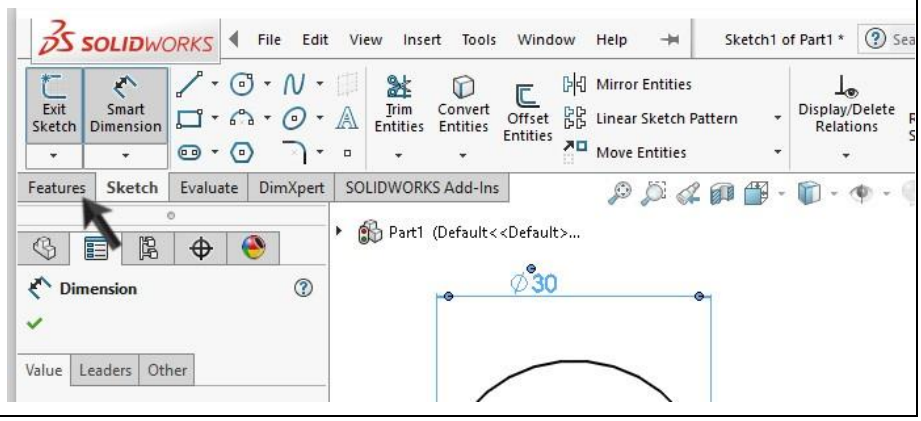
1	Startup SOLIDWORKS. To do so, find SOLIDWORKS in the Start-menu of Windows. There even may be a shortcut at your desktop that you can use. After start-up, you will see an image as printed at the right side of this page. The screen may look a bit different; this depends on the default settings of the software and/or the computer you are using.	
2	No file has been opened yet: to do so, click on the first button at the toolbar: 'New'.	
3	Next a new screen appears: (see right image) Click on 'Part' and then 'OK'.	

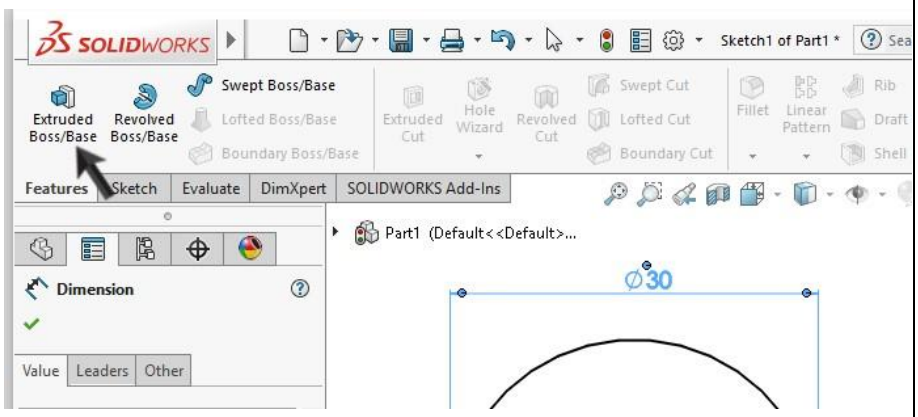
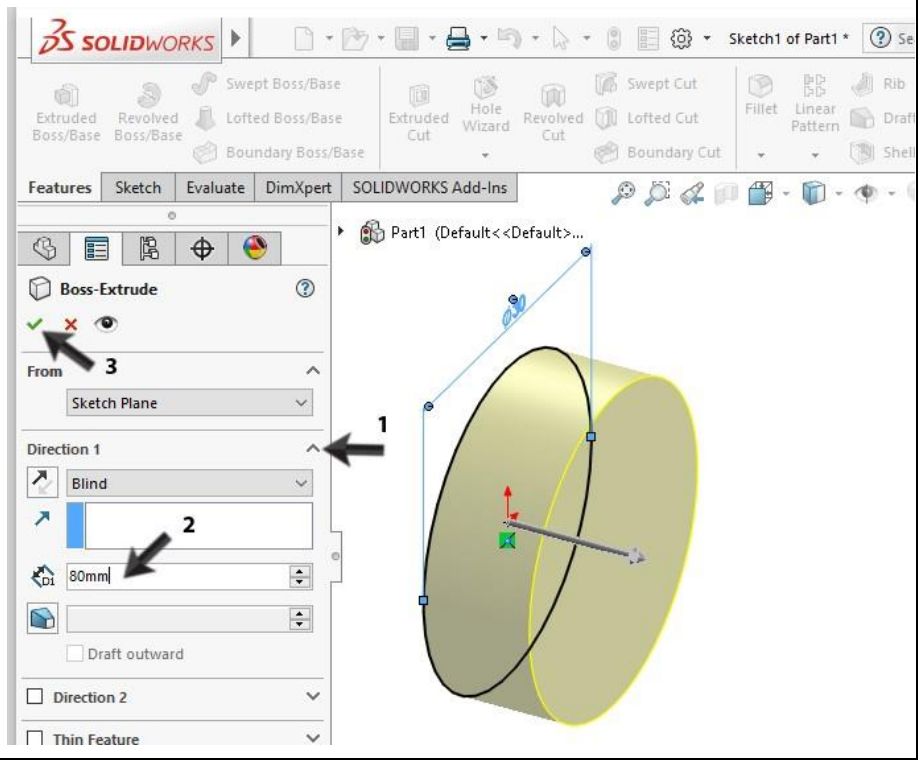
4	In the left column, click on 'Right Plane'. The plane turns blue. On this plane we are going to make a drawing.	
5	Click on 'Sketch'. New buttons appear and you can use them to make a drawing.	
6	Click on 'Circle', in order to draw a circle.	

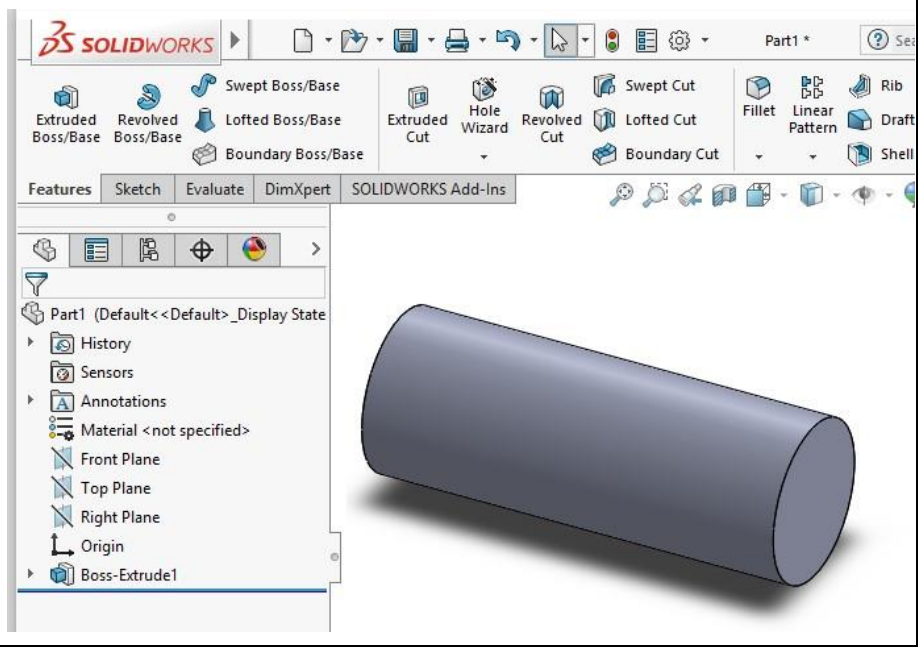
7	At this point, the plane turns towards you, so you can have a good view on what you are drawing. In the middle you see some red arrows; this is what is called the 'origin' or the zero markers. Put the cursor directly at the origin: it should look like the image on the right. Click once with the left mouse button.	
8	Move the cursor away from the origin. The radius of the circle will appear close to the cursor. Make sure this radius is approximately 15. When the cursor is at the right position, click again to draw the circle.	

9	Next we will add a dimension. Click on 'Smart Dimension'	
10	1. Click somewhere on the circle. 2. Next move the mouse up and click again to add the dimension above the circle.	

11	A small menu appears automatically in which you can change the dimension to the desired value. Change the dimension into 30 and click on OK (the green 'OK' sign)	
	Tip!	Would you like to change a dimension after you finished drawing? Doubleclick on the dimension. The menu will re-appear and you can change the value.

12	The drawing (Sketch) is ready now and we can use it to make a threedimensional shape. Click on 'Features' at the top of the screen. The function buttons to create three-dimensional shapes appear.	

13	Click on 'Extruded Boss/Base'. With this feature you will add material.	
14	By doing so, the drawing revolves, so you get a good look on what you are doing. At the left of the screen a number of fields appear, either open or closed. 1. Be sure the field 'Direction 1' is opened. If not, click on the double arrows next to the field title. 2. Enter a length of 80. 3. Click OK.	

15	Congratulations! You've created your first shape: an axis! In SOLIDWORKS, we call this a 'Feature'.	

Tip!

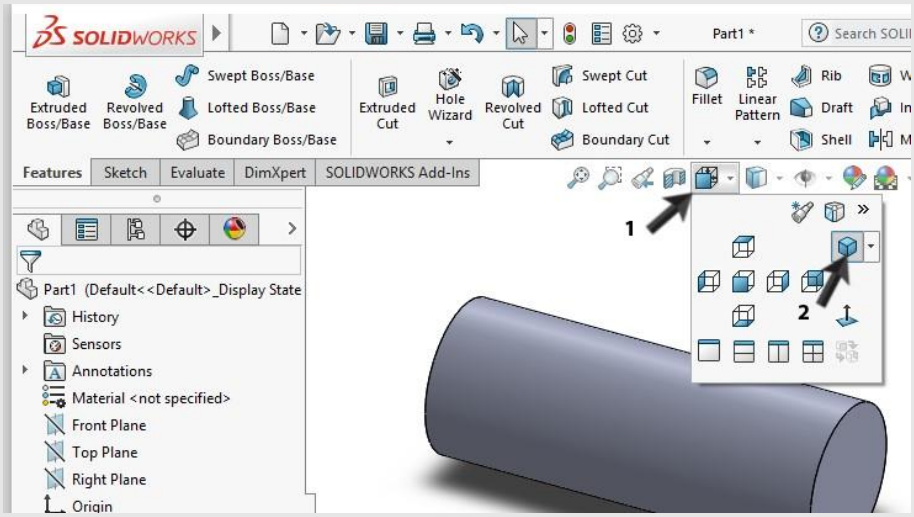
Sometimes the part you have created does not fit the screen OR you want to view it from another side. In SOLIDWORKS you only need the scrollwheel from your mouse.

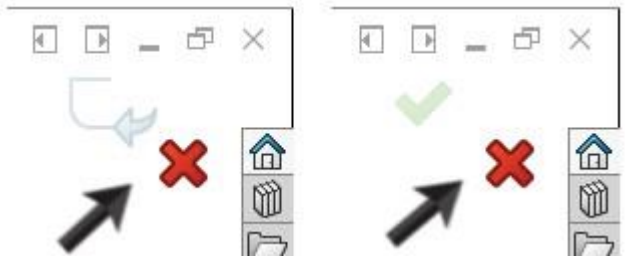
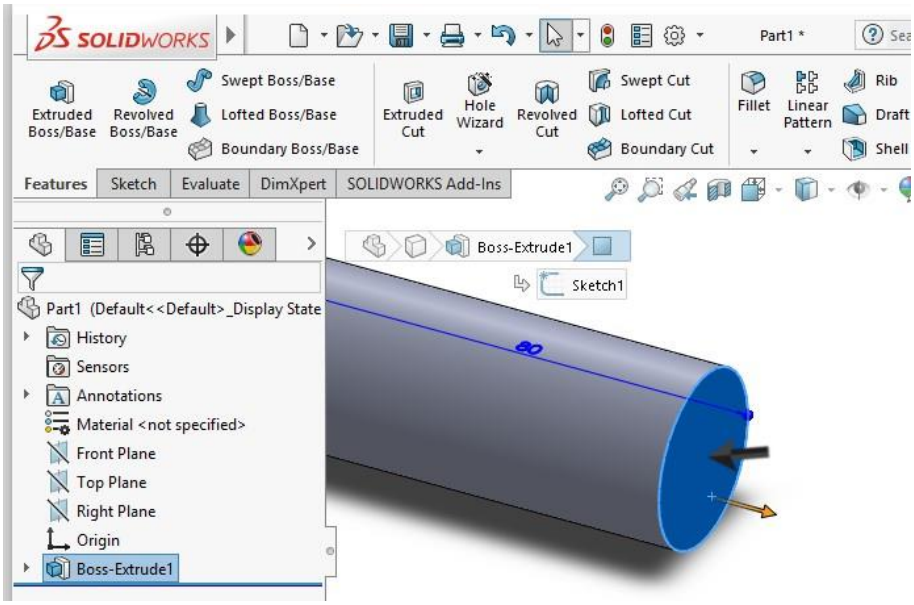
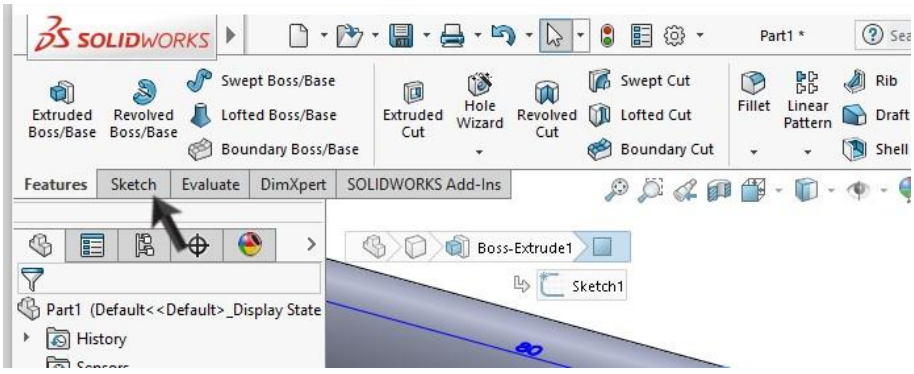
To zoom in- or out: **turn** the scroll-wheel. The position of the cursor determines at which position you are zooming.

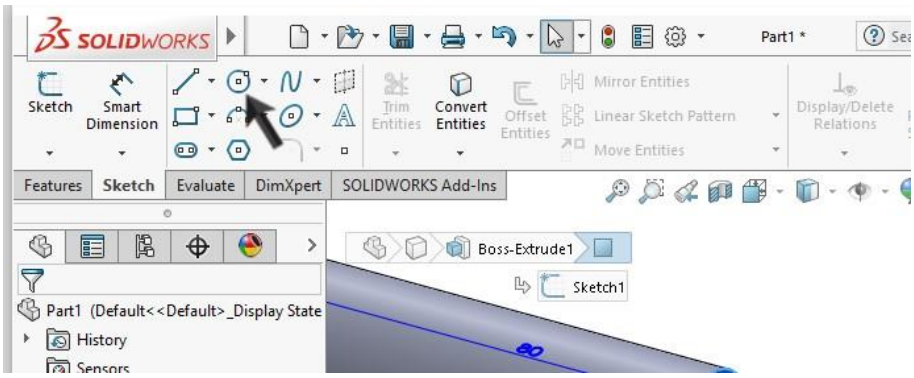
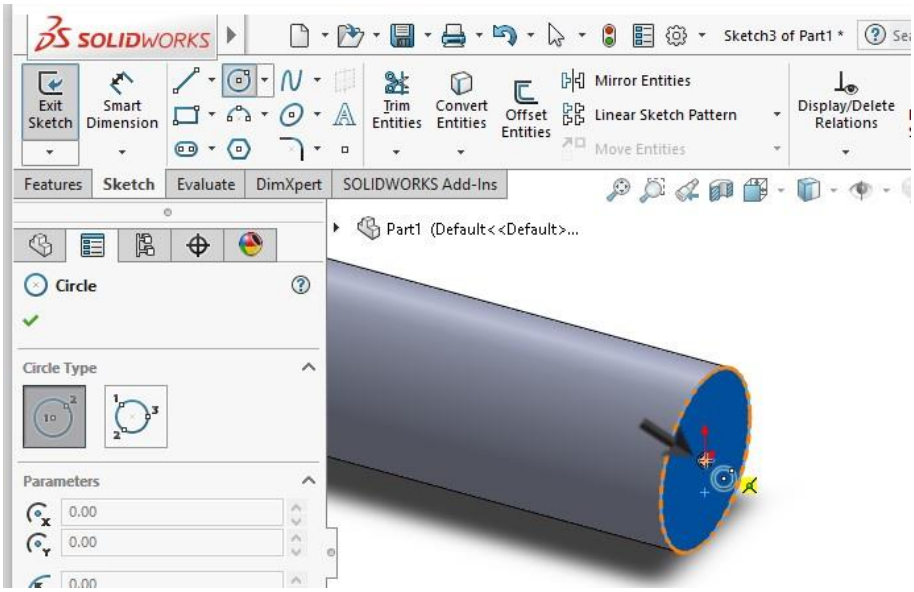
To rotate your part: **press the scroll-wheel** and move your mouse.

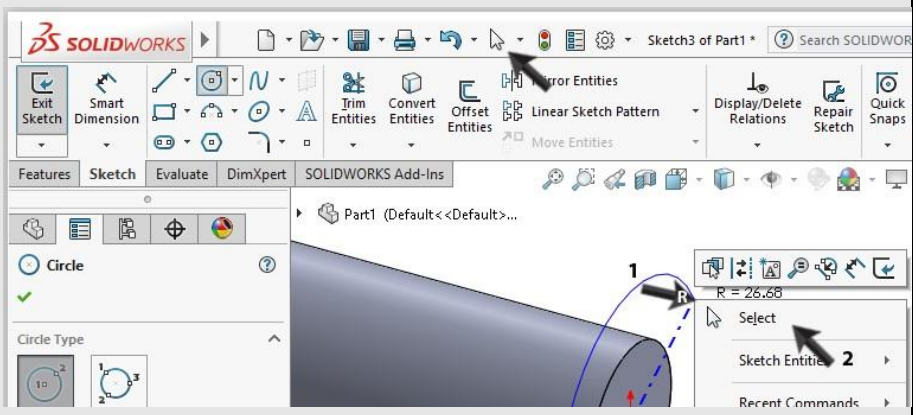
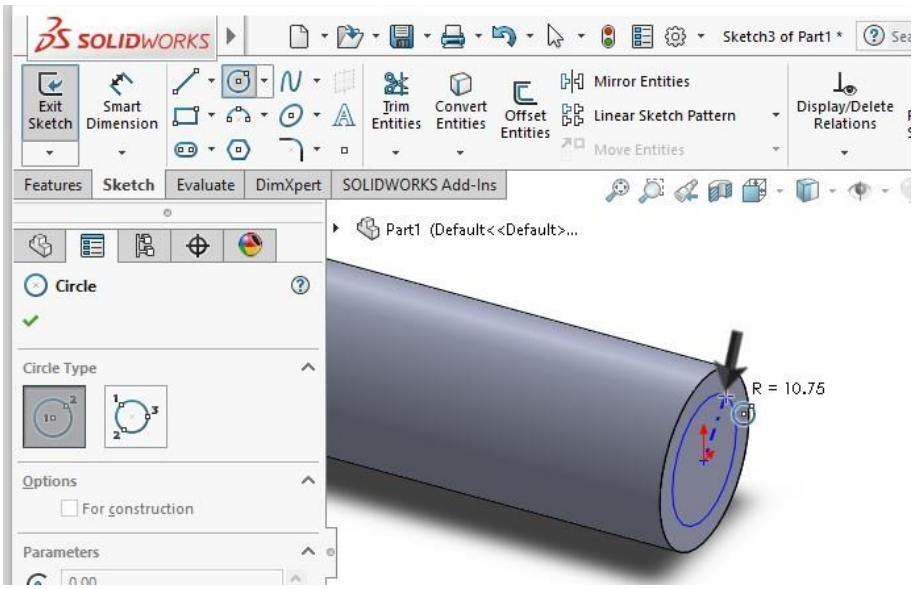
You may need some practice to get the part in the desired position. If you get lost completely, you can get back to the default position:

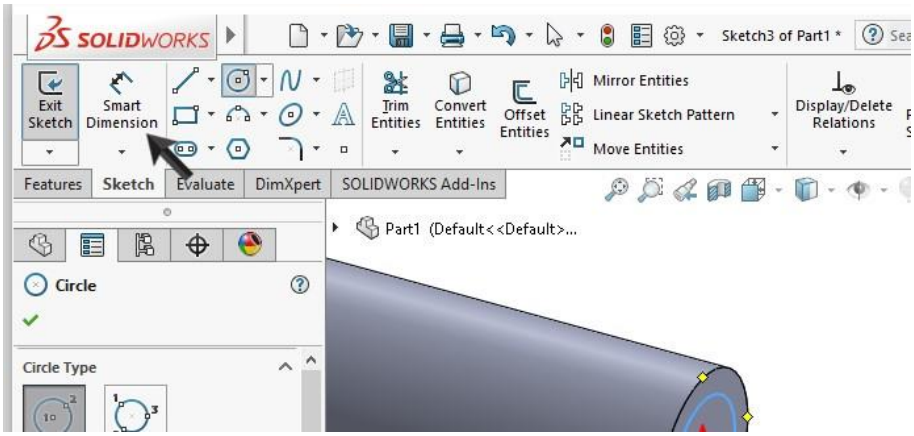
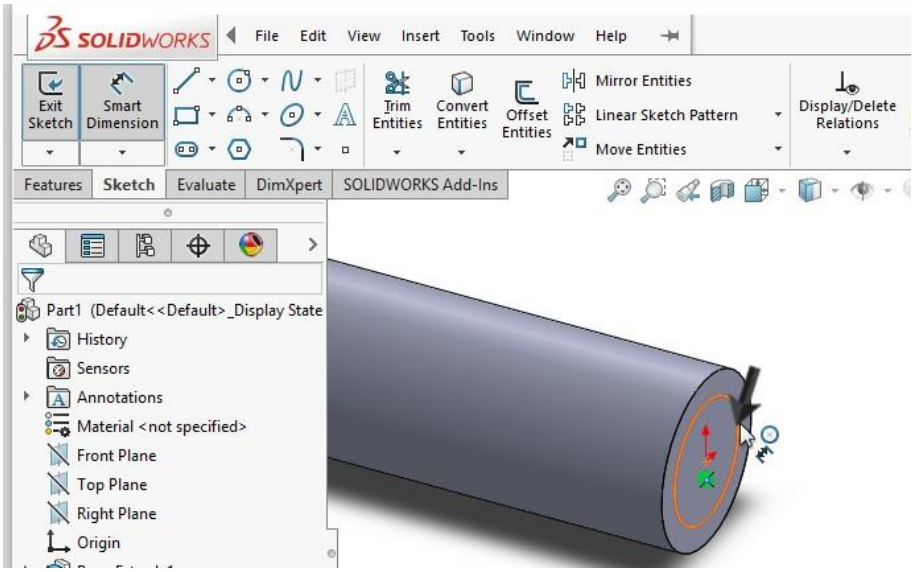
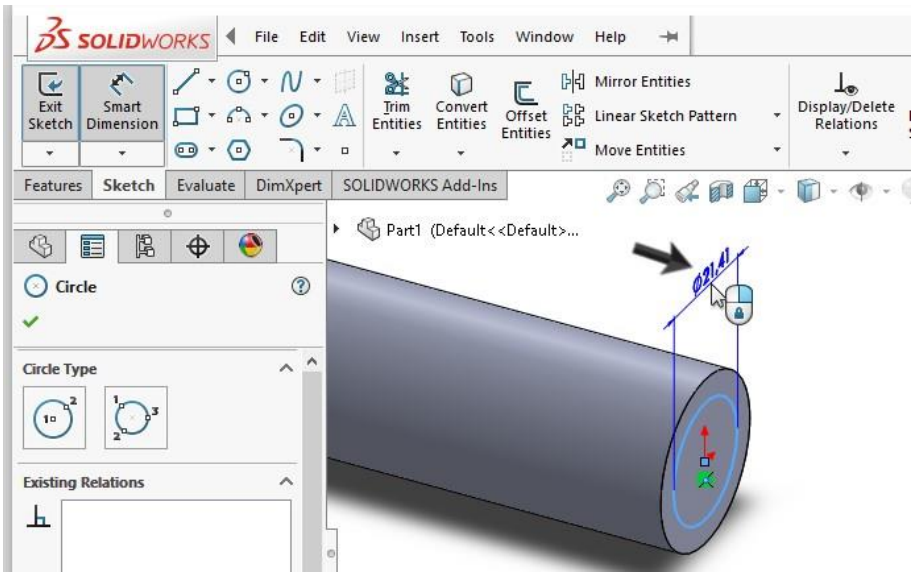
1. Click on View Orientation at the top of the screen.
2. Choose one of the default view orientations

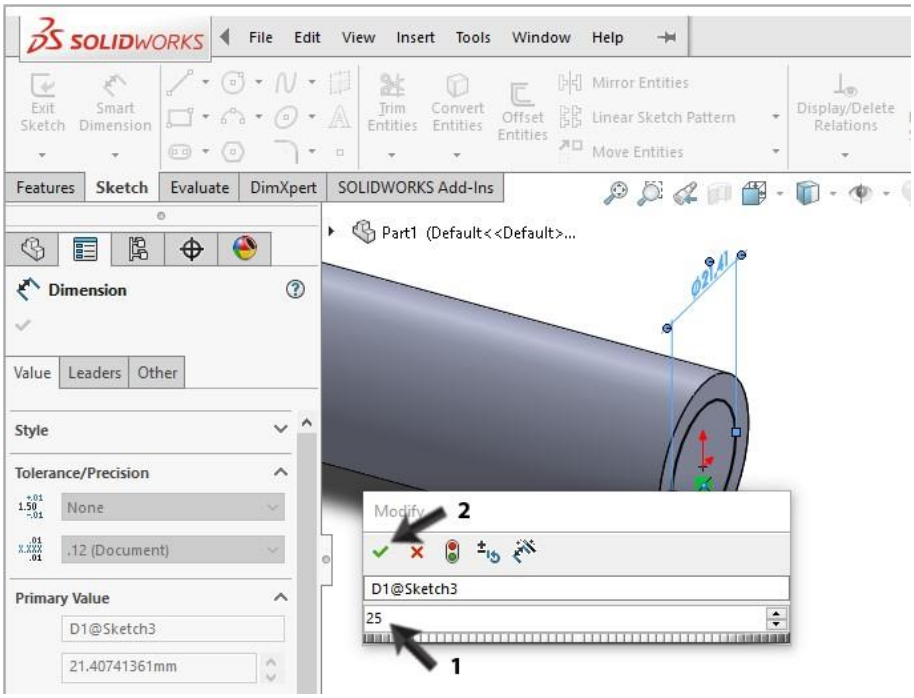
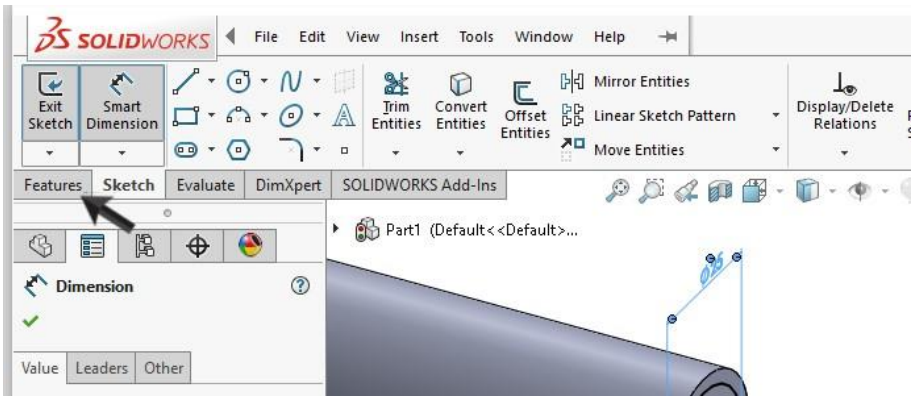
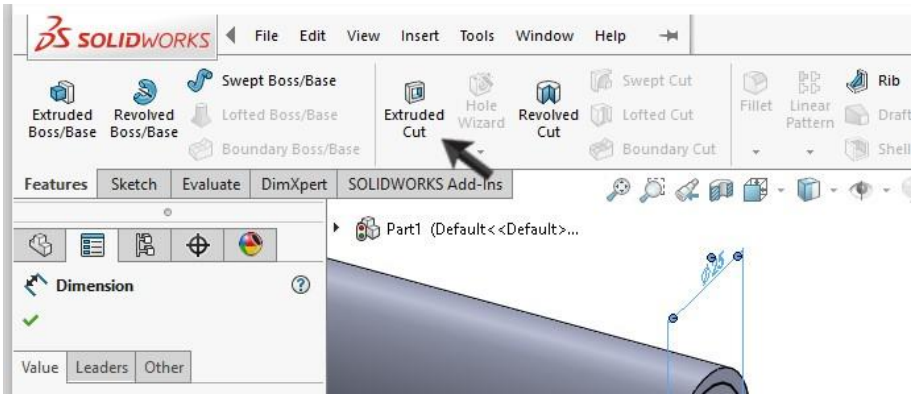


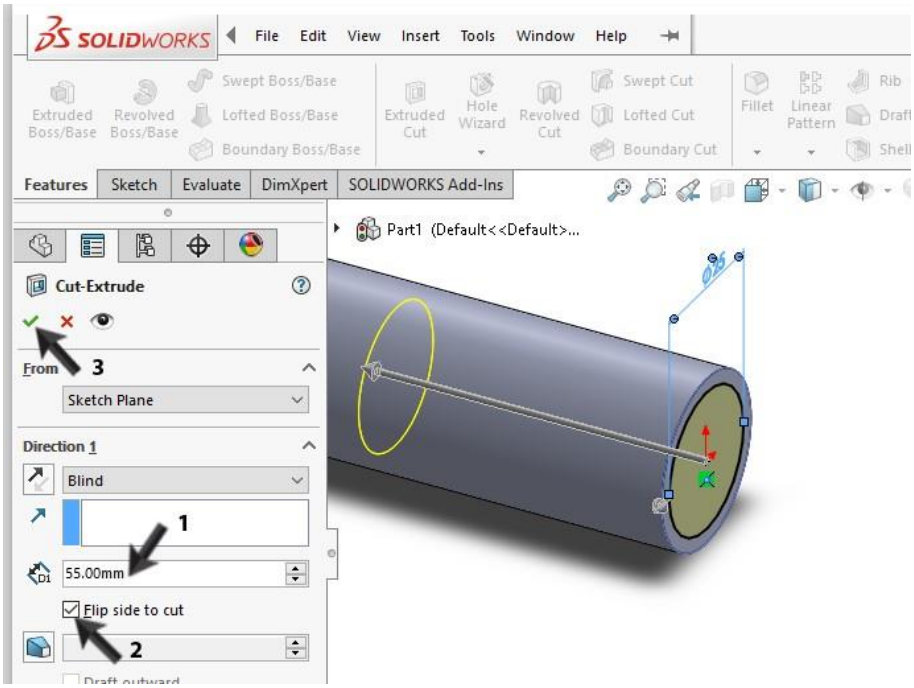
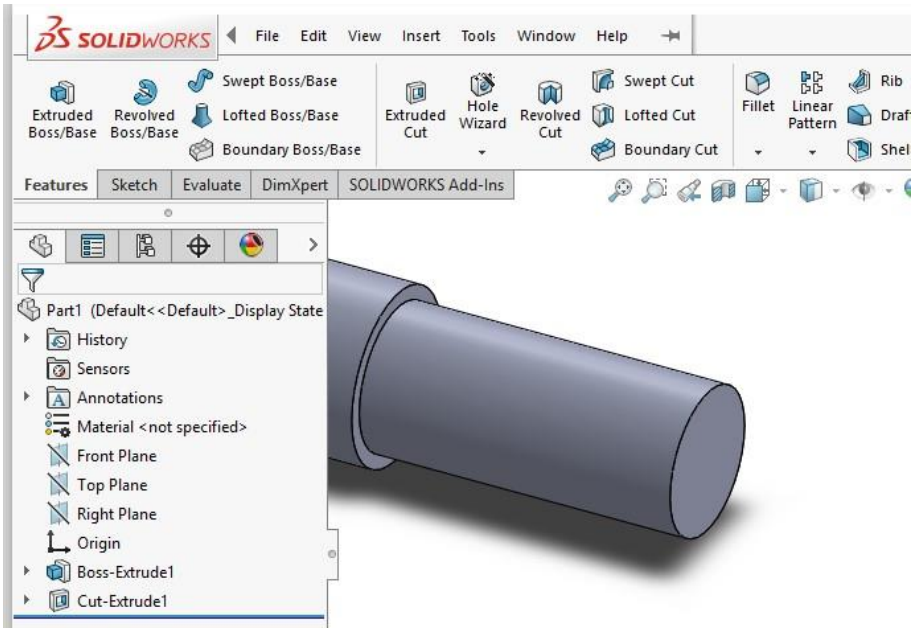
16	Next we are going to make a new feature, but before you do, it is very important to check if all other actions are completely finished. Watch the right upper corner of the screen. Does is look like the image on the right? This means that you have not entirely finished the last action. Click on the red cross to close the last command, only then you can start a new one!	
17	Next we are going to change the diameter. Click on the top plane of the axis to select is. Be sure not to select the edge, but the plane! When you do this right, the plane turns blue.	
18	Click on Sketch to show the sketch-commands.	

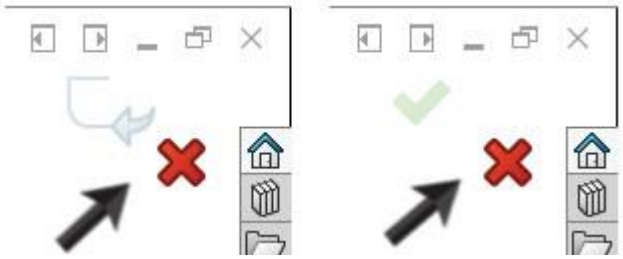
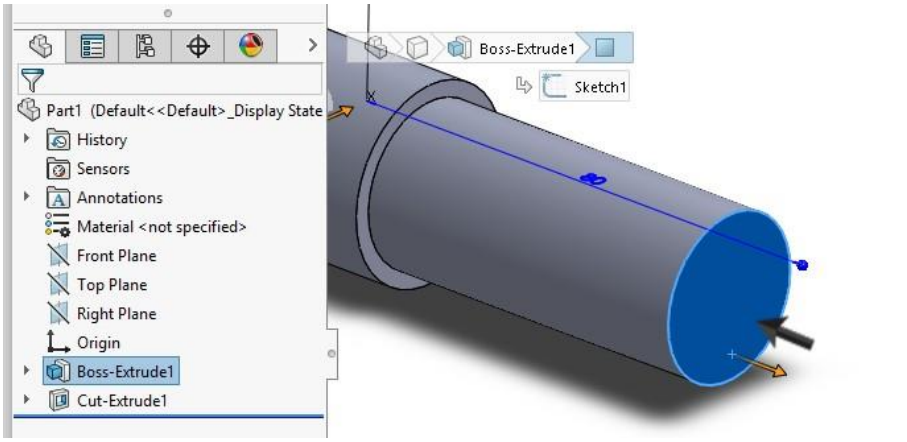
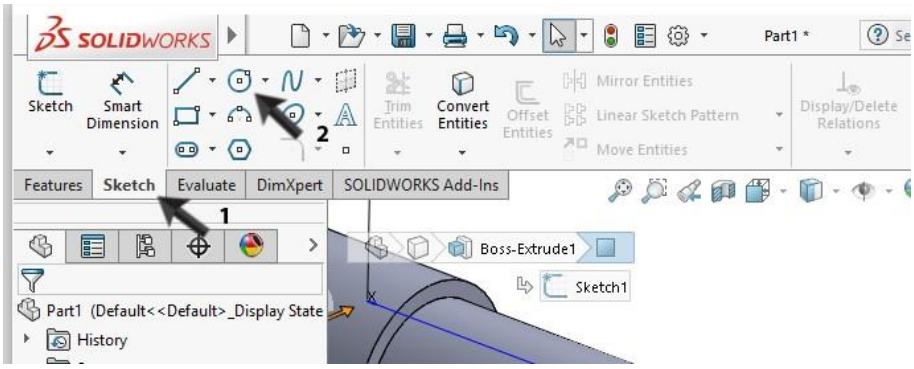
19	Click on 'Circle'	
	Tip!	If you cannot get a clear view on what you are doing, zoom in or rotate your part. Remember: • To zoom in- or out: **turn** the scroll-wheel. The position of the cursor determines at which position you are zooming. • To rotate: **press the scroll-wheel** and move your mouse.
20	Point the cursor at the centre of the circle. If you move the cursor over the center of the circle (without clicking) the cursor changes like in the right image. Click only when the cursor has the right shape, or else you will fail to select the right item.	

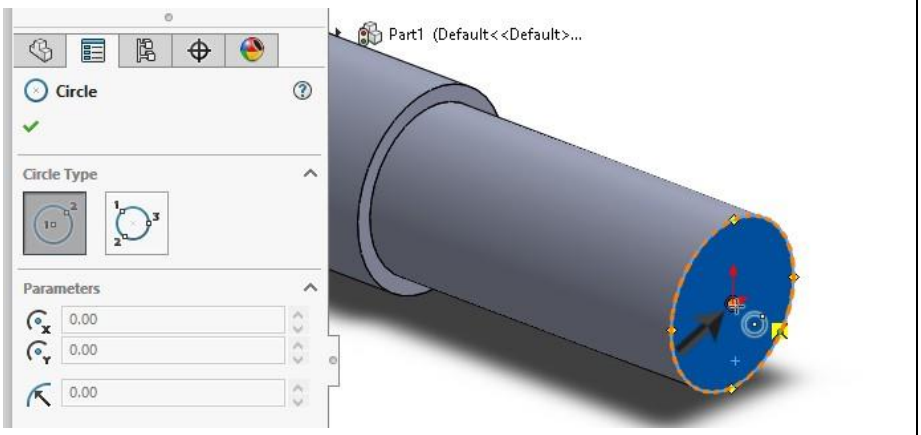
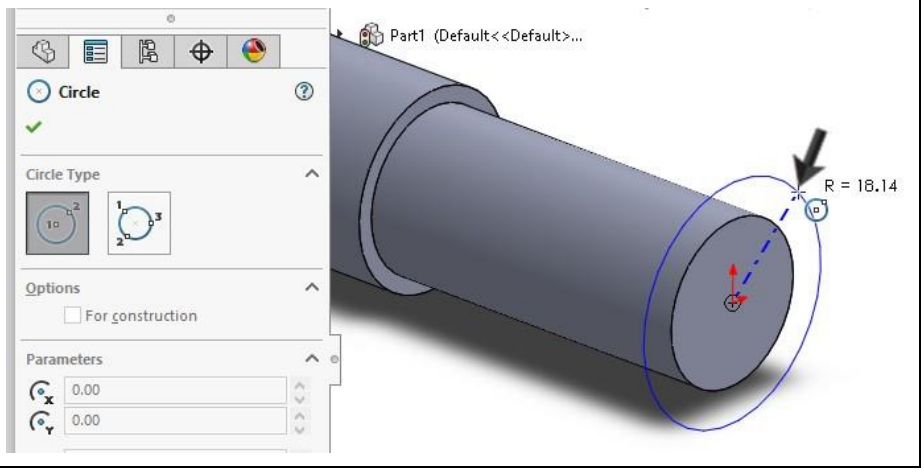
	Tip!	Did you chose a wrong item anyway or do you want to abort a command, press the <Esc> on your keyboard. You can also click the right mouse button and chose 'Select' in the menu that appears. When the command is aborted you can start another one or throw away an element if you want to do so: Click on the element in de sketch, and press <Delete> at your keyboard. (Pay attention: do NOT use the <Backspace>button!). 
21	Move the cursor away from the centre and click at any point to draw the circle. The dimension does not matter yet. **Pay attention: do NOT click on** another element like the outer circle of the plane.	

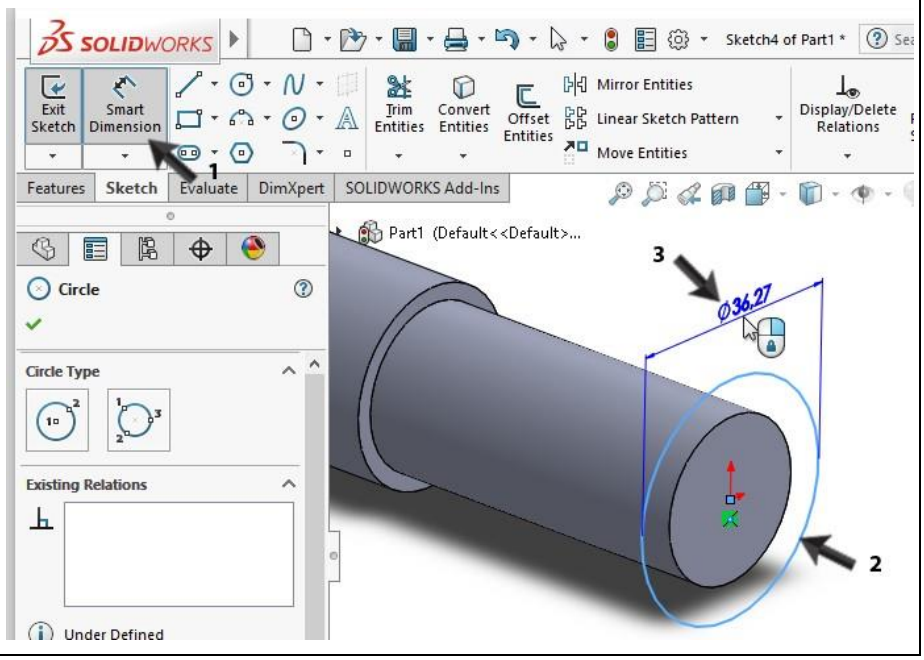
22	Click on Smart Dimensions.	
23	You have just drawn a circle. Next click on it.	
24	Move the cursor away from the circle and determine a position to put down the dimension. **Pay attention: do NOT click on** another element, because SOLIDWORKS will than calculate the distance between the circle and that element instead of displaying the diameter of the circle!	

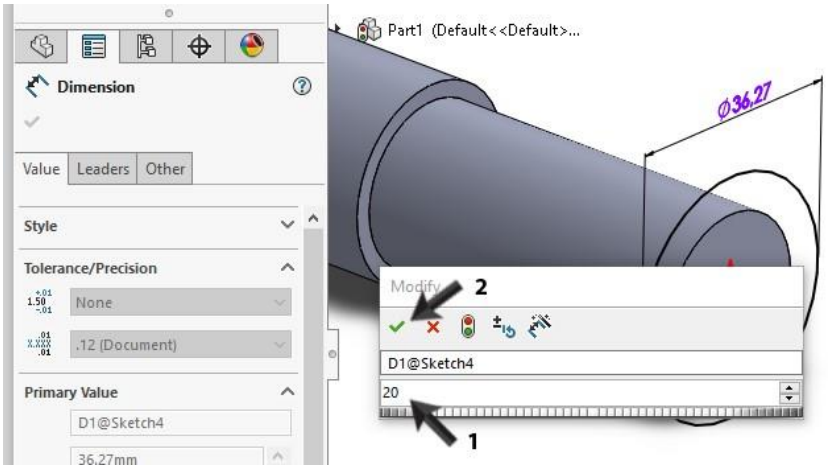
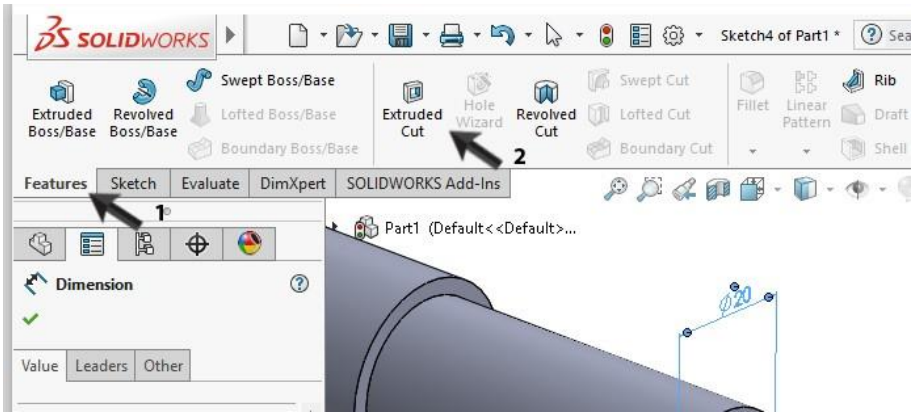
25	A menu appears in which you can change the dimension. Change it to 25 and click on OK.	
26	Click on Features to show the functions to add or remove material.	
27	Click on Extruded Cut. With this command we can remove material.	

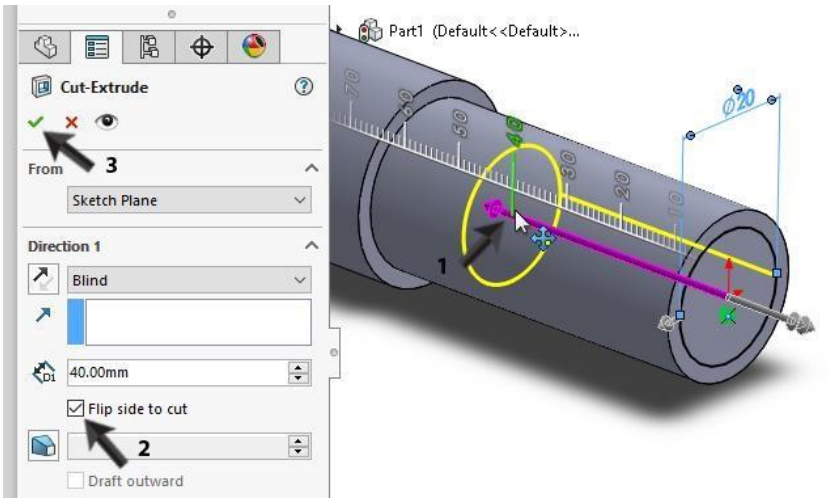
28	In the PropertyManager set the following: 1. A depth of 55 2. Mark 'Flip side to cut'. This makes sure, that the material on the **outside** of the circle and not at the **inside** of the circle is removed. 3. Click OK.	
29	The first cut is made! We will make the second cut in exactly the same way. We will speed up through the steps now.	

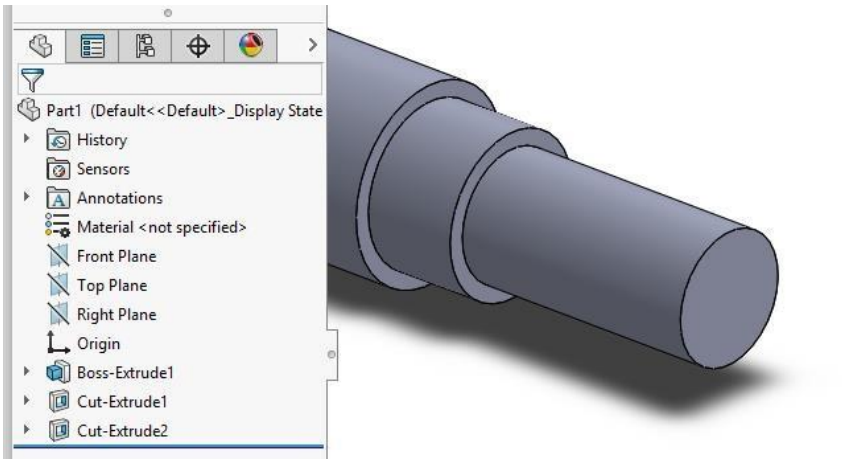
30	Before you start a next feature check if there is there is another command or sketch active. Check the right upper corner of the drawing field. When a red cross like in the right image is visible, click on it to close the last command.	
31	Select the end of the axis. Be sure to select the plane and not the edge!	
32	Click on 'Sketch' first (to show the right functions) and after that click on Circle.	

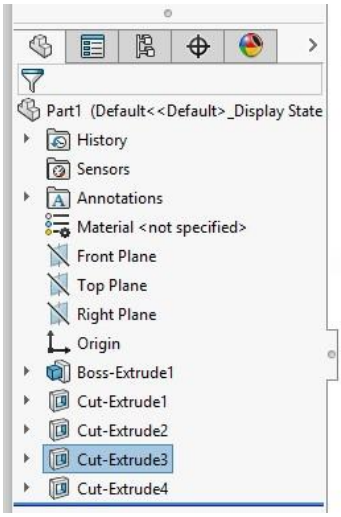
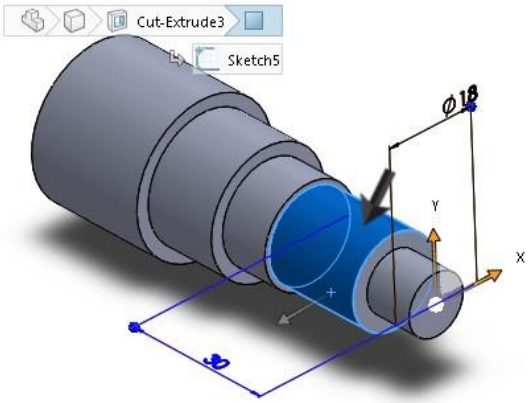
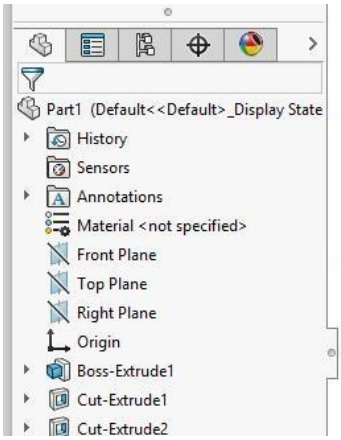
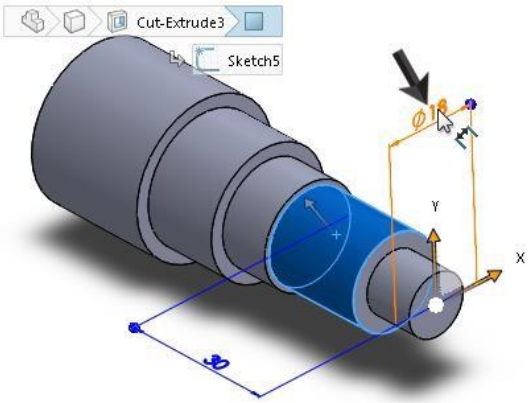
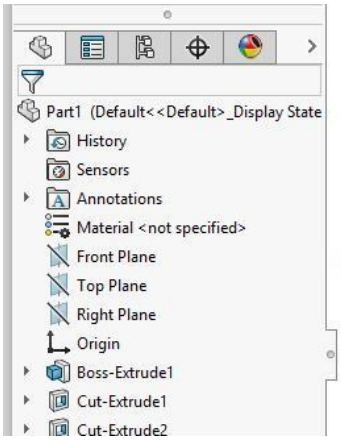
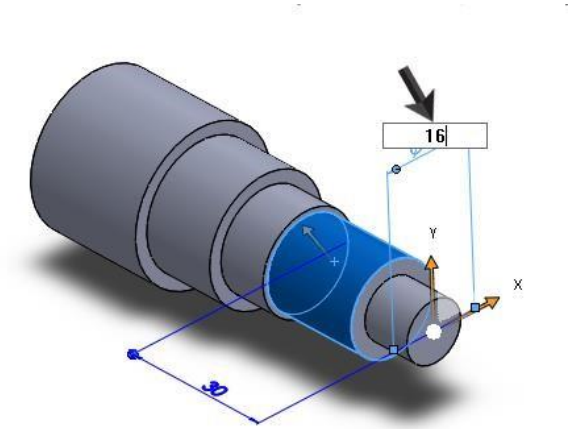
33	Click on the centre of the axis. Notice the shape of the cursor!	
34	Click somewhere outside the material to draw a circle.	

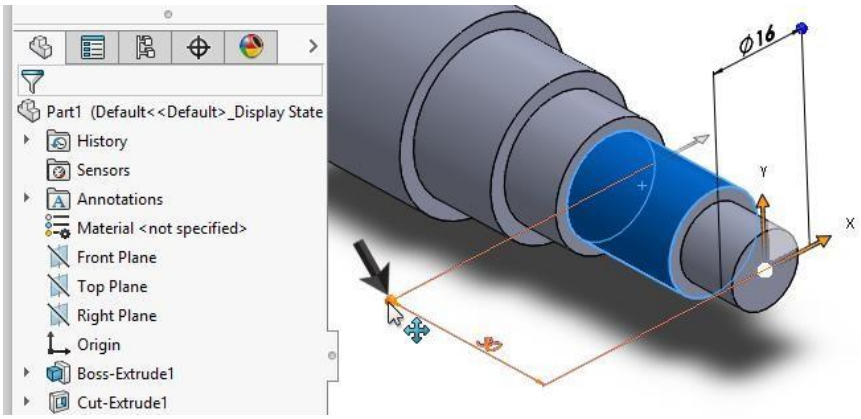
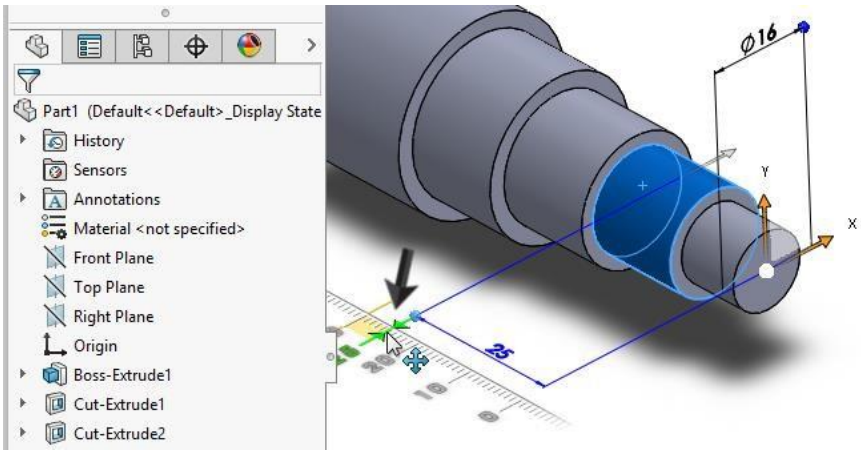
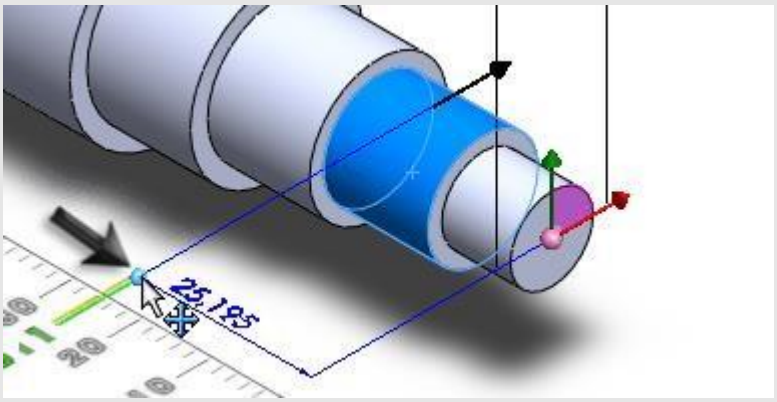
35	Next add a dimension to the circle: 1. Click on 'Smart Dimensions'. 2. Click on the circle (it turns blue, remember?) 3. Click above the part (**do not click on another element)** to position the dimension.	

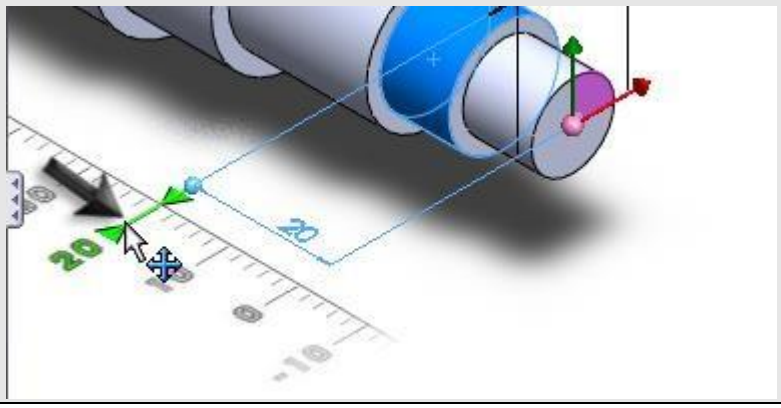
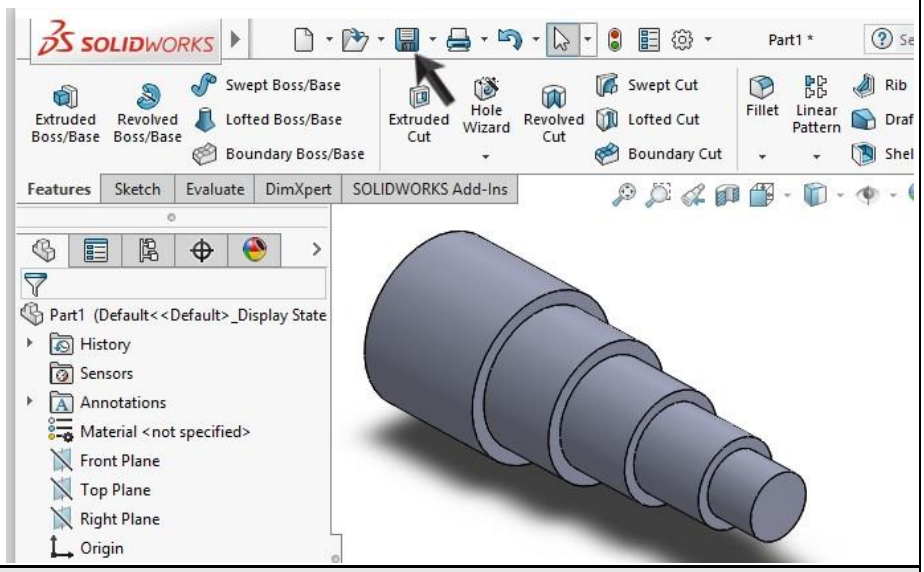
36	Change the dimension to 20 and click OK.	
37	Click on 'Features' to show the right functions and next on 'Extruded Cut' to remove material.	

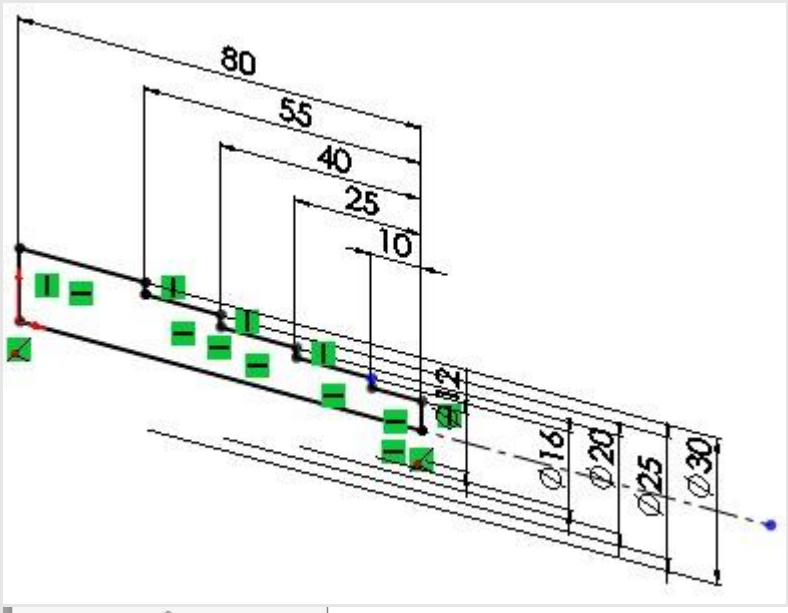
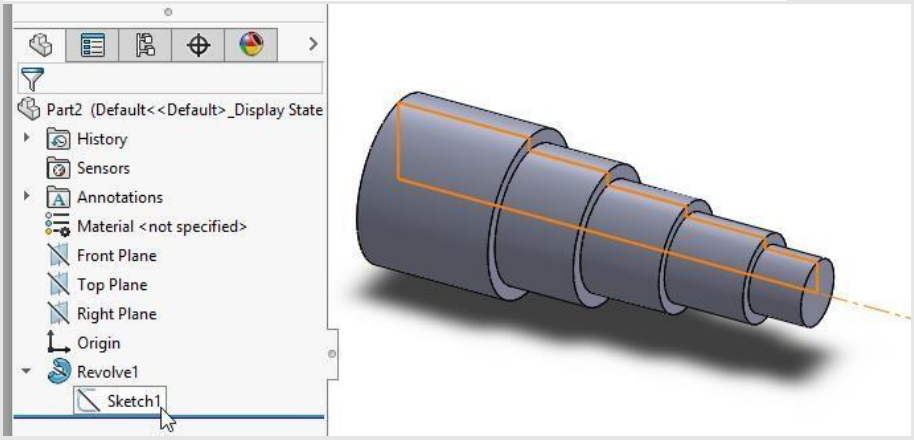
38	Next set the following: 1. Set the depth on 40 by dragging the arrow in the part. As soon as you start dragging a ruler appears. Release the mouse button as soon as the dimension reads 40. 2. Mark 'Flip side to cut' 3. Click on OK.	

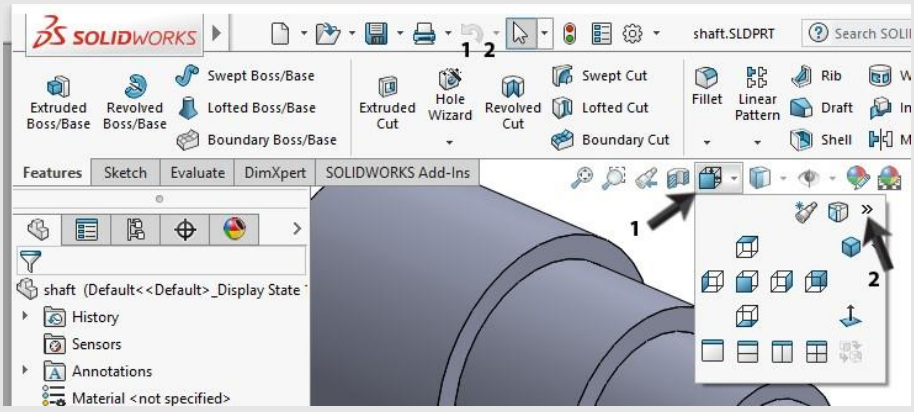
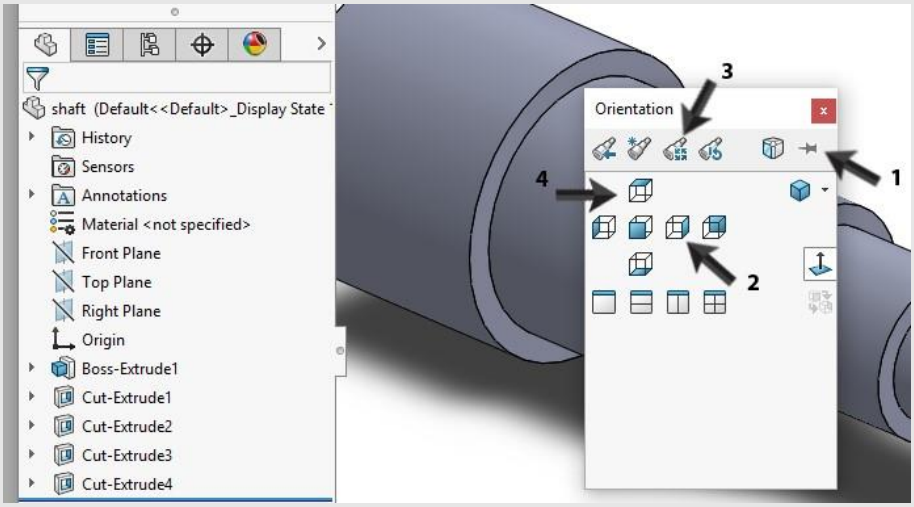
	Tip!	Until now you have seen two ways to set the depth of an extrusion: 1. You can enter the dimension in the field at the left of the screen. You did so at step 14 and 28. 2. You can drag the arrow in the part, as you did in the last step. Choose for yourself what you think of as the best way.
39	The second cut is made!	
	Finish the model!	Two other cuts have to be made at exactly the same way. Only the dimensions are different now: • The third cut has a diameter of 18 and a length of 30. • The fourth cut has a diameter of 12 and a length of 10. Follow the same steps as we did before: 1. Check if no commando is active. 2. Select the plane of the axis. 3. Draw a circle and set the right diameter 4. Make an Extruded Cut to remove material.

40	We now see that the dimensions of the third cut are wrong! It says Ø18x30, but it should be: Ø16x25. How do we adjust this? In SOLIDWORKS you will find it's very easy to do! Click in the part on the third cut. In the part the dimensions will appear: Ø18 and 30. NOTE: make sure that the command Instant3D is turned on in the CommandManager, otherwise things might work a bit different. 	 
41	First we adjust the dimension of Ø18. Click on this dimension once.	 
42	A small menu appears in which you can change the dimension. Enter: 16 and press <Enter> at your keyboard. The part changes immediately to its new dimension.	 

43	You can also change the length of 30 in the same way, but we will show you now that you can also change this by dragging it. At the left hand of the dimensions you will notice a small blue sphere. Click on it in order to drag it.	
44	You will notice that the ruler appears and you can drag it to a dimension of 25.	
	Tip!	Watch where the cursor is while dragging: - Is the cursor **next to the rules,** you are dragging in a random way and you will never get an exact dimension of 25 mm.  - Is the cursor pointing at the ruler, you can make an accurate change. Zoom in if your ruler is not accurate enough.

		
45	We have now changed the length AND the diameter of the third cut. Fantastic! The first part is finished now! Click on 'Save' in the toolbar and name the part axis.sldprt.	
	What are the most important items you have learned so far?	This first exercise was to get to know SOLIDWORKS. You have learned a few things you must remember very well: • Extruding means you can add or remove material. 1. Use Extruded Boss/Base to add material. 2. Use Extruded Cut to remove material. • To make a shape or part you almost always do this in two steps: 1. Draw a sketch: create a two-dimensional drawing in a plane. 2. Make a feature: you create a three-dimensional shape. • Before you want to start a new feature, be sure no other command is active and no sketch is still open. • You can easily adjust all dimensions. How to make more complicated adjustments, we will show you in one of the next tutorials.

	Is there another way to create this part?	Sure! Most parts you create with SOLIDWORKS can be created in several ways. You cannot say there is a 'good' or a 'bad' way to do so. In this exercise we have used the way you would make this part on a lathe in the workshop. This is often a good guideline for building a part. In this case for instance you could have also drawn the contour of the part and rotate it afterwards. In a next exercise we will look into this method in detail.  
	Would you like to rotate the coordinate system of the axis?	In step 4 of this tutorial we chose to make the first sketch on the Right plane. That couses the axis to be modeled horizontal. Usualy this is just fine, but sometimes in the end it turns out that that wasn't the best choice. For example if you would like to use the model in a CNC-program. Then the axis should be oriented vertically. With the next steps you can change the orientation afterwards.

	1. Click on View Orientation 2. In the menu that appears, click on the two arrows to see more options.	
	1. Pinpoint the menu, so you can give multiple commands. 2. Click on 'Right', the top of the axis will rotate towards you. 3. Click on Update Standard Views. 4. Click on Top. 5. SOLIDWORKS will ask for a confirmation: click 'Yes'. Now you have changed the position of the axis in the space. If you click on Iso again, the axis will stand up right.	

Thank you please follow for more tutorials.