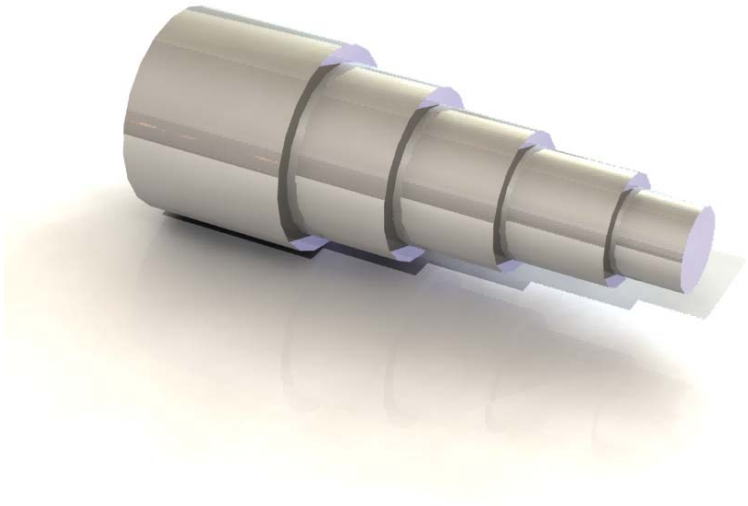
SolidWorks® Tutorial 1

Axis



Preparatory Vocational Training
and Advanced Vocational Training

Stichting Consortium Beroepsonderwijs

SolidWorks

platform metaal & metalektro

To be used with SolidWorks® Educational Edition Release 2008-2009

SolidWorks Benelux developed this tutorial for self-training with the SolidWorks 3D CAD program. **Any other use of this tutorial or parts of it is prohibited.** For questions, please contact SolidWorks Benelux. Contact information is printed on the last page of this tutorial.

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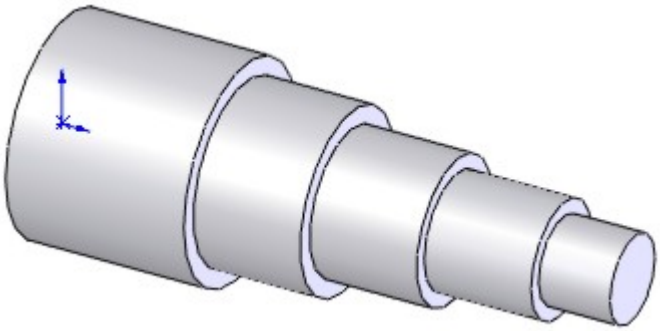
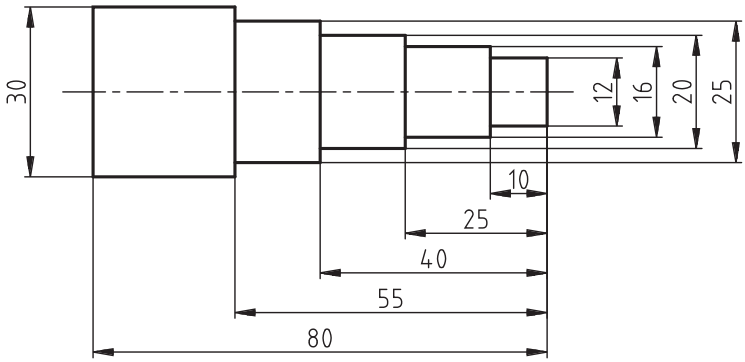
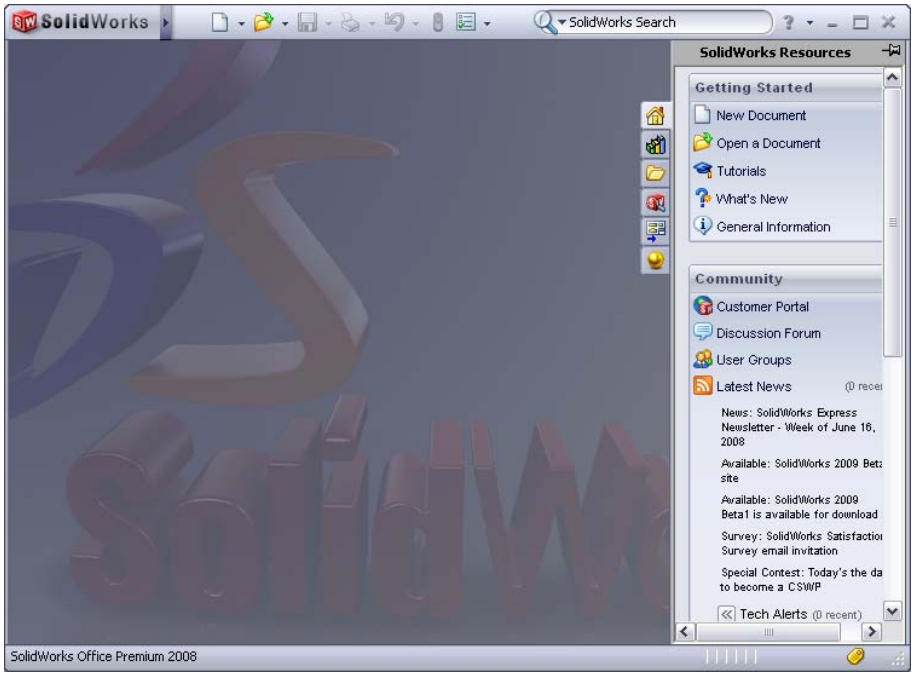
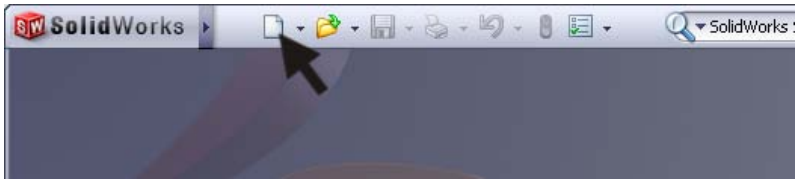
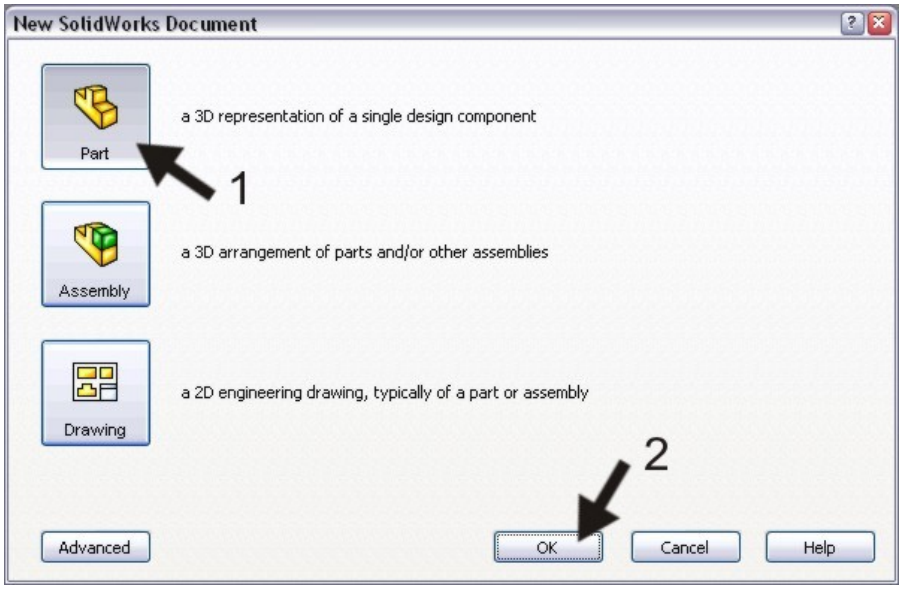
Axis

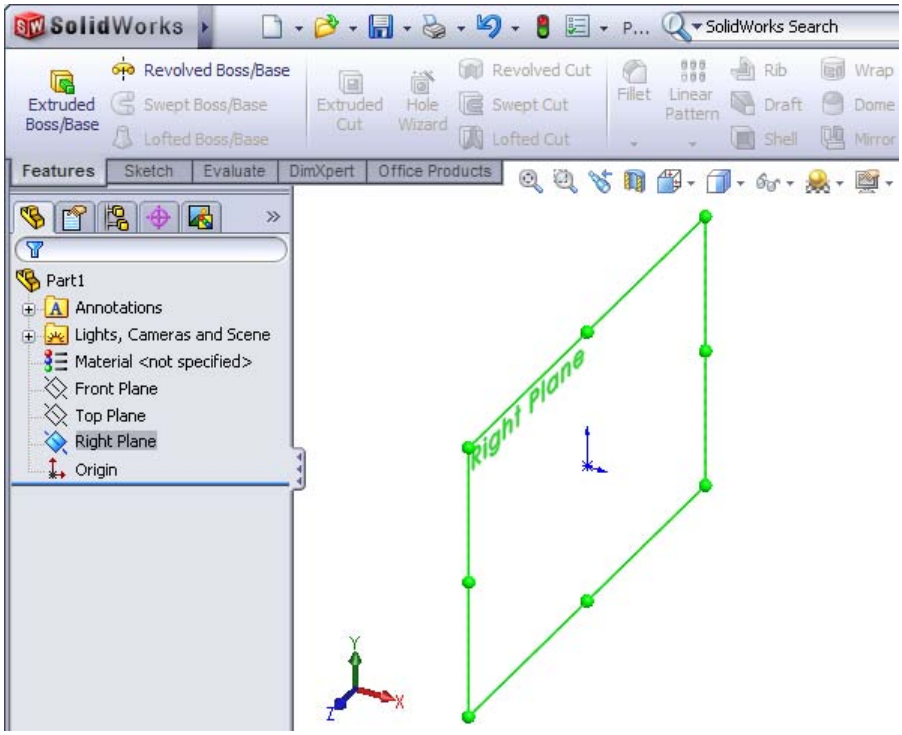
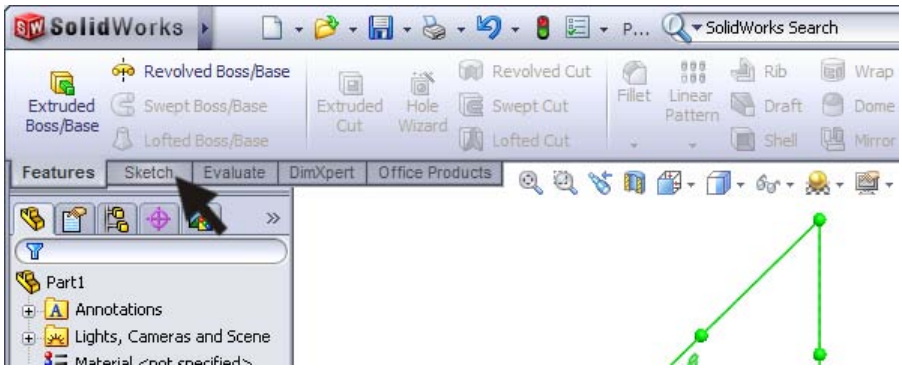
This first exercise provides an introduction to SolidWorks software. First, we will design and draw a simple part: an axis with different diameters. You will learn how to work with the software and learn its basic principles. You will find out how to add and remove material.

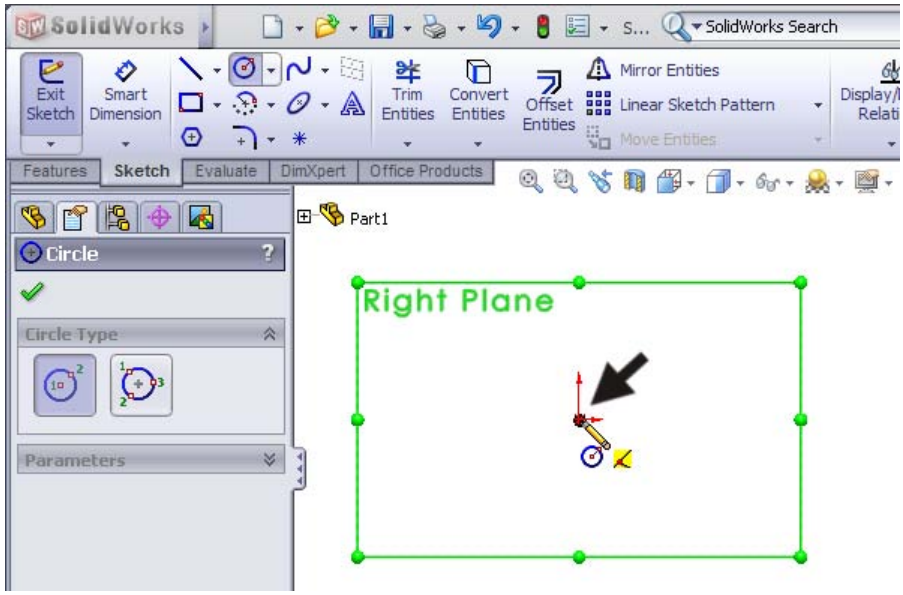
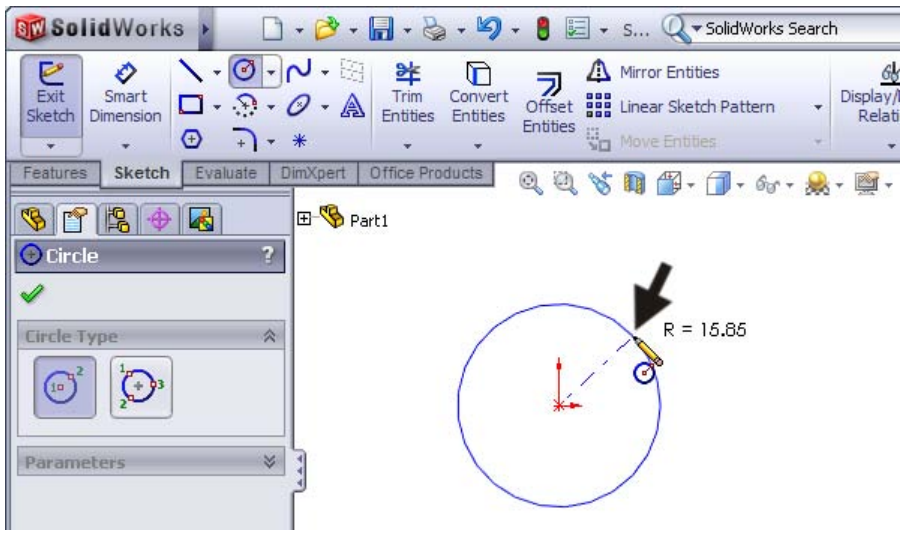
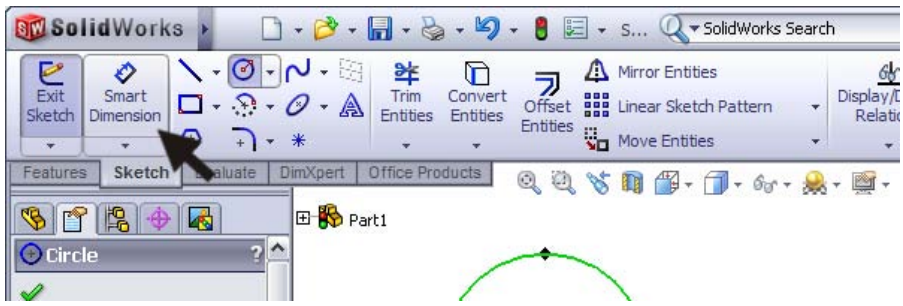


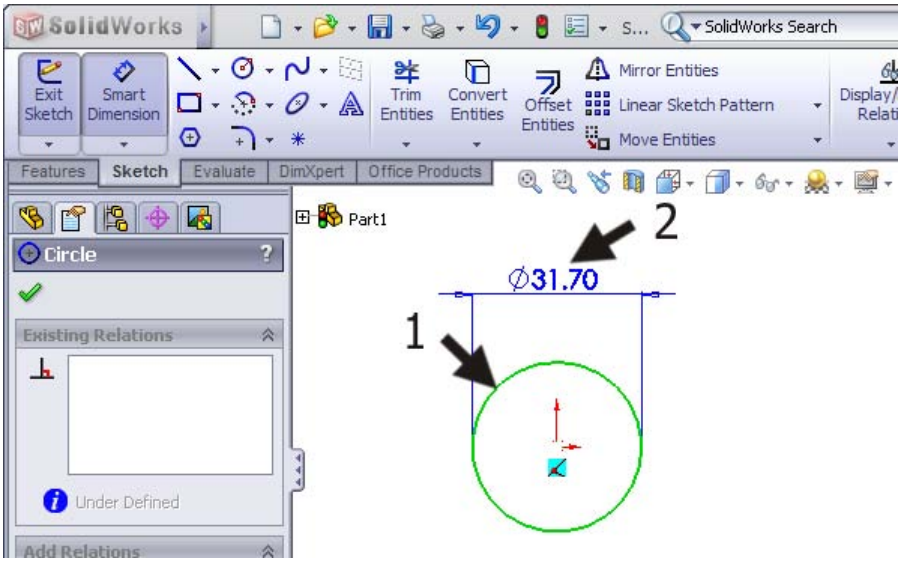
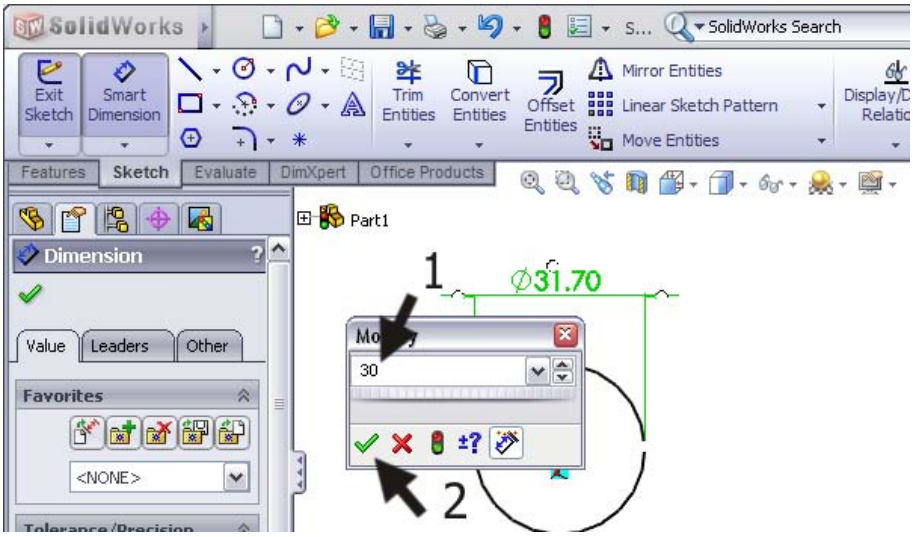
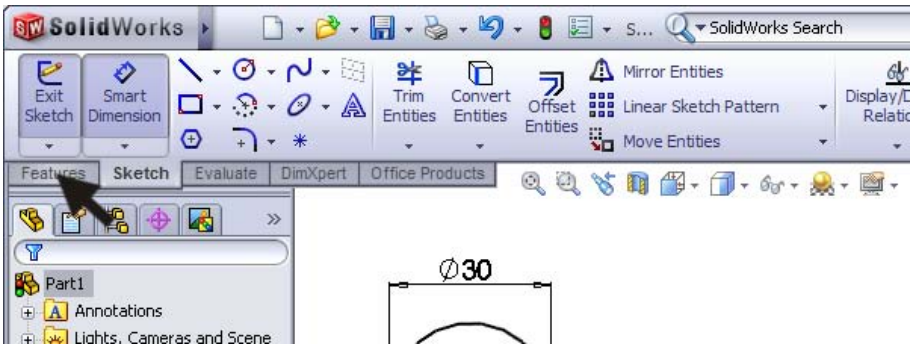


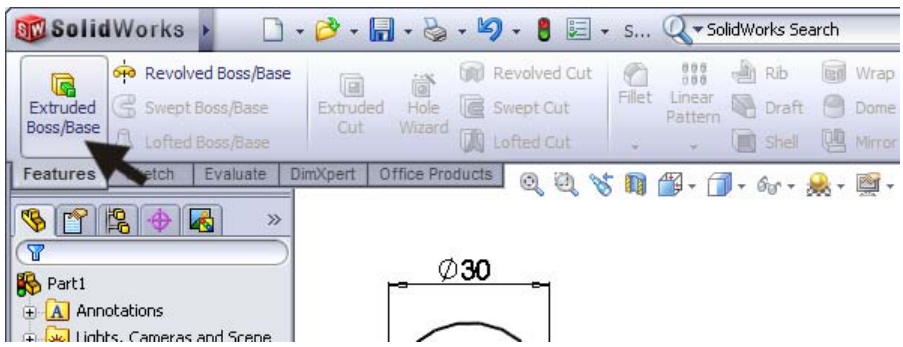
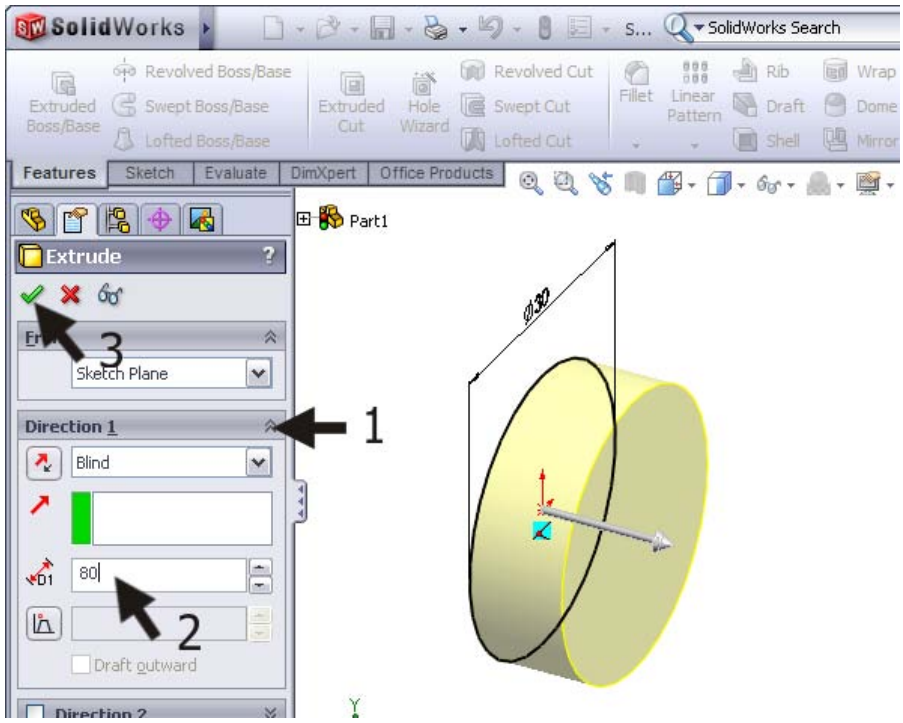
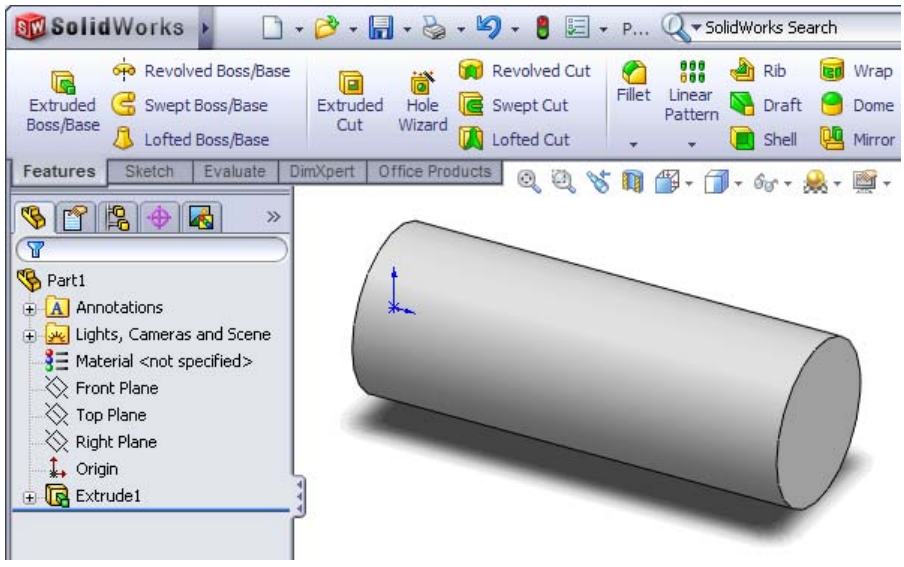
How to do it	Before you start drawing in SolidWorks, you must have a work plan of how to proceed. In most instances, you will produce a part in SolidWorks in the same way as you would create it in the workshop. Therefore, for this assignment you have to go through the following steps: 1. Create an axis of Ø30 x 80. 2. Cut the material in order to create the different diameters. At the turning machine, you would have to perform several extra steps to achieve the desired accuracy. For example, you would not be able to remove all the material in a single turn. In SolidWorks, this is not the case.

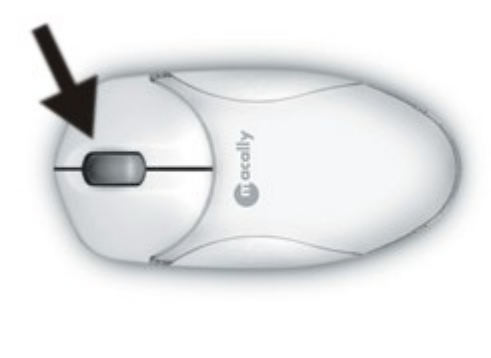
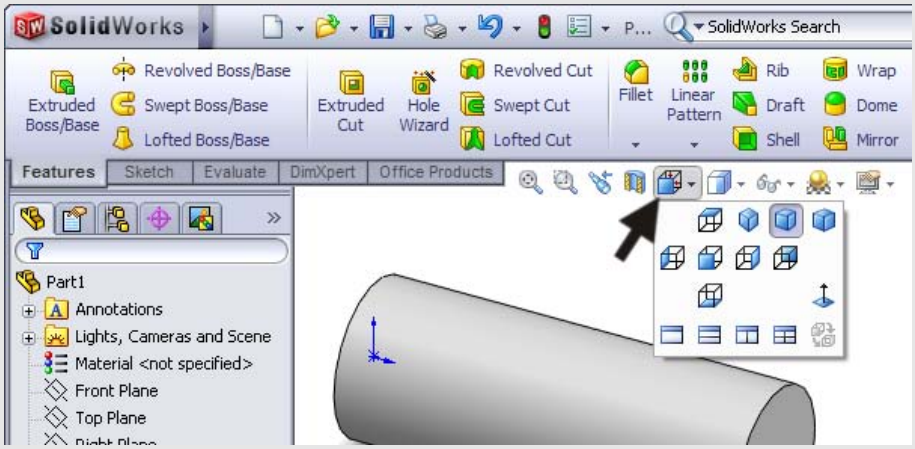
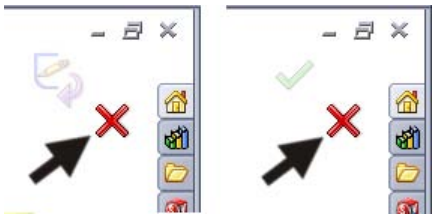
1	Start up SolidWorks. Do this by locating SolidWorks in the Windows Start menu of. There may even be a shortcut on your desktop that you can use. After startup, you will see an image like the one at the right side of this page. The screen may look a bit different; this depends on the default settings of the software and/or the computer you are using. 	
2	No file has been opened yet. To create a file, click on the first button on the toolbar: 'New'.	
3	Next, you will see a new screen (see right image). Click on 'Part' and then click 'OK'.	

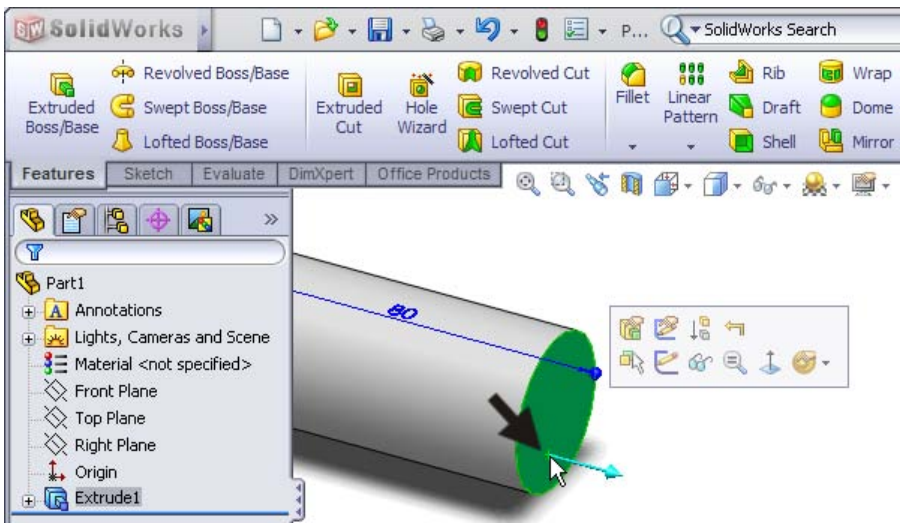
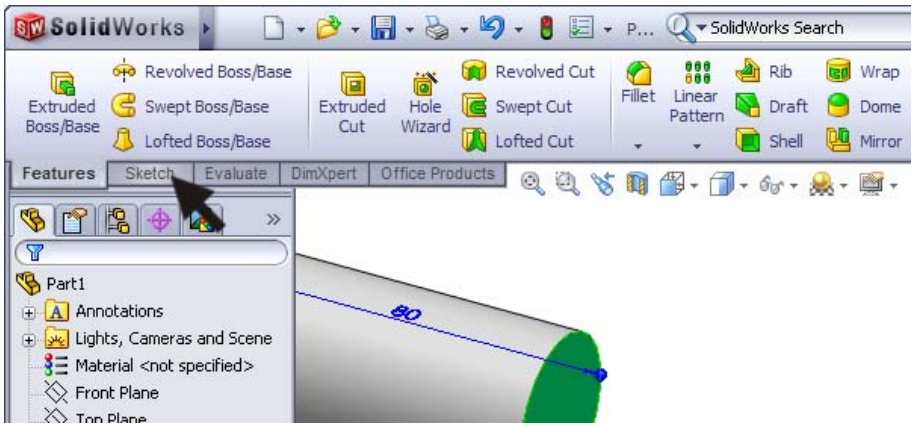
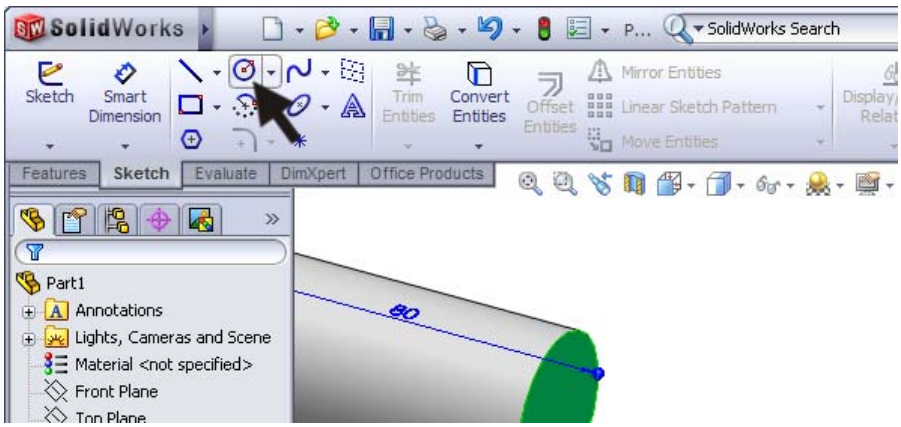
4	In the left column, click on 'Right Plane'. The plane turns green: We will make a drawing in this plane.	
5	Click on 'Sketch'. New functions and possibilities appear, and you can use them to make a drawing.	
6	Click on 'Circle', in order to draw a circle.	

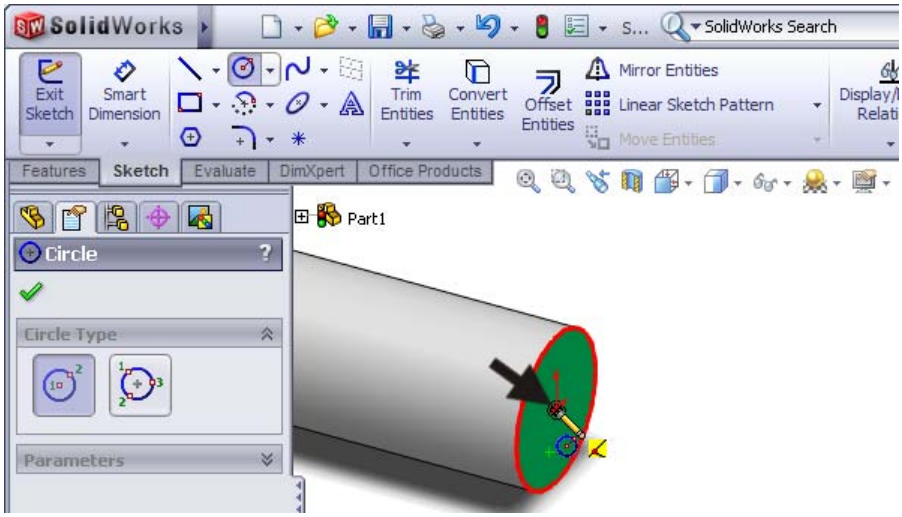
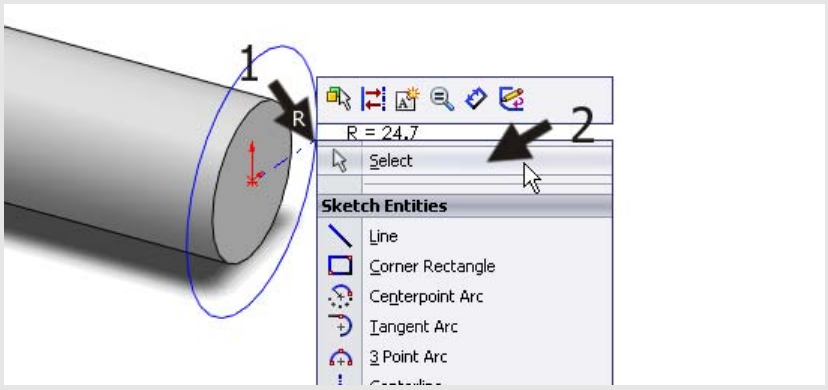
7	At this point, the plane turns towards you, so you can have a good view on what you are drawing. In the middle you see a point with red arrows; this is what is called the 'origin' or the zero marker. Put the cursor directly at the origin: it should look like the image on the right. Click once with 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'.	

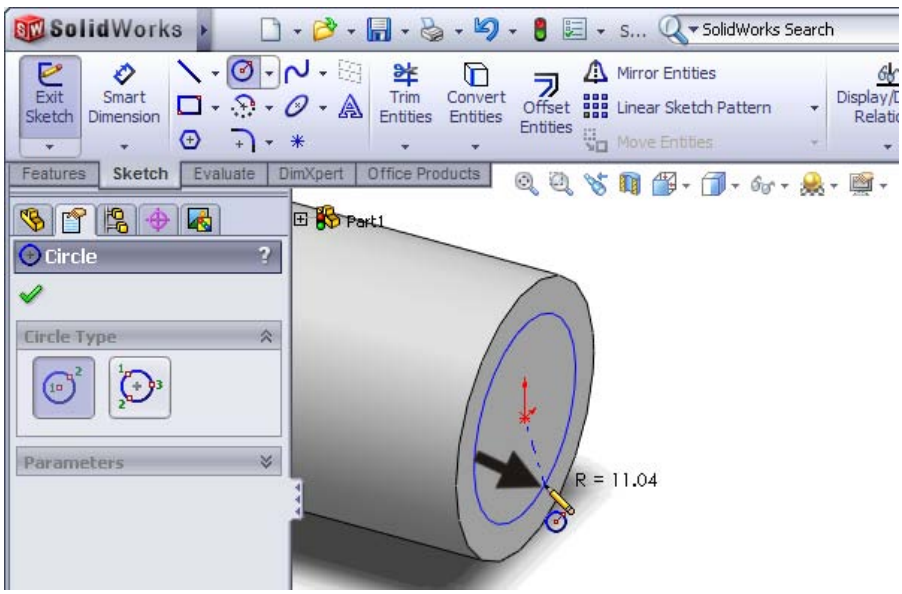
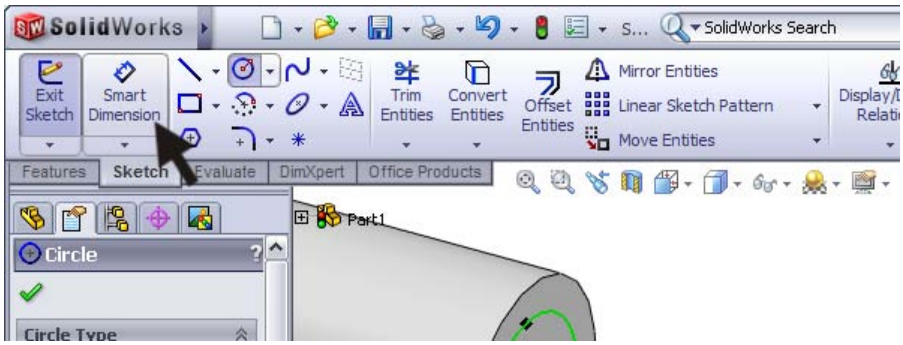
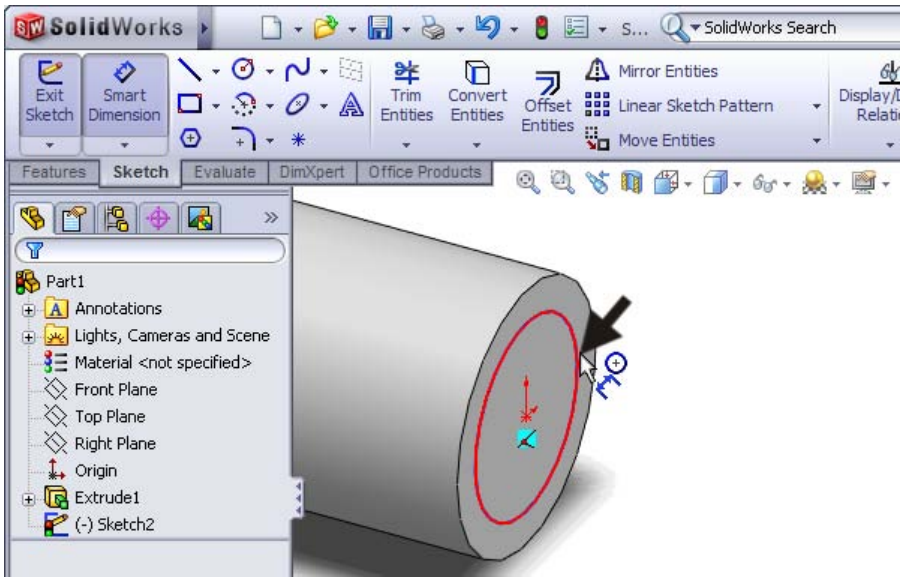
10	Click on any point on the circle. Next, move the mouse and click again to add the dimension above the circle or at the position you want it to be.	
11	A small menu automatically appears through which you can change the dimension to the desired value. Change the dimension to 30 and click on OK (the green 'OK' icon).	
	Tip!	Would you like to change a dimension after you have finished drawing? Double-click on the dimension. The menu will reappear and you can change the dimension.
12	The drawing (Sketch) is now ready, and we can use it to make a three-dimensional shape. Click on 'Features' at the top of the screen. The function buttons needed to create three-dimensional shapes appear.	

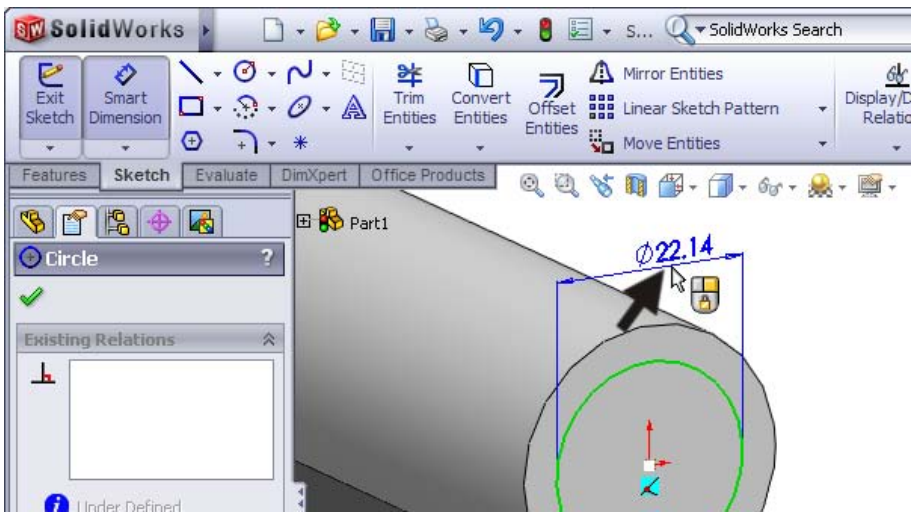
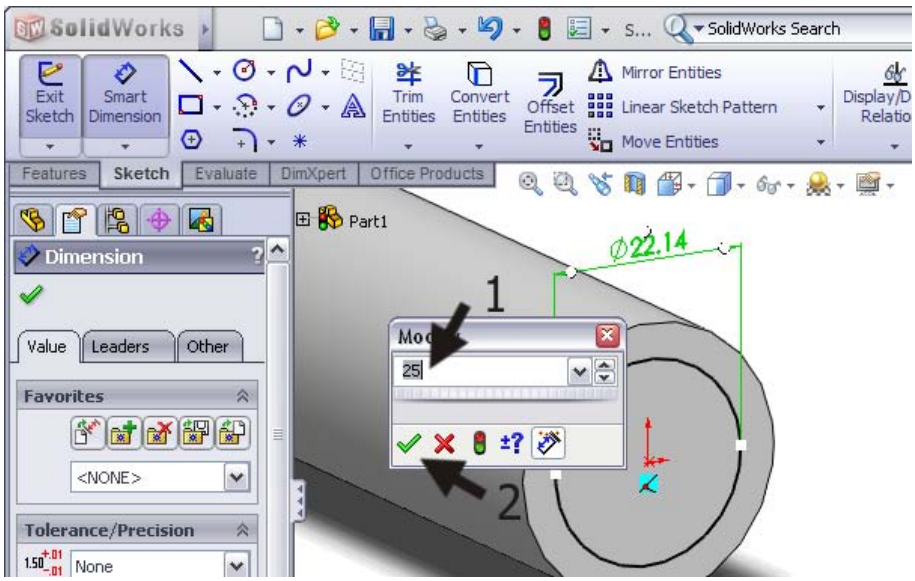
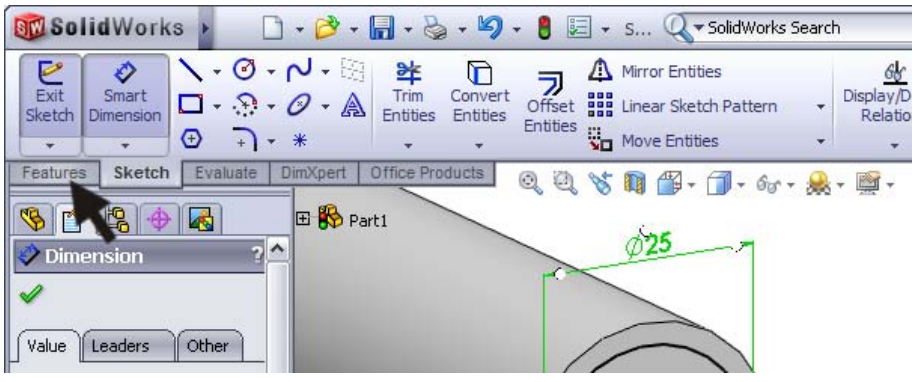
13	Click on 'Extruded Boss/Base'. You will add material with this feature.	
14	When using this tool, the drawing revolves so you get a good look at what you are doing. A number of fields appear at the left of the screen, either open or closed. Be sure the field 'Direction 1' is opened. If not, click on the double arrows next to the field title. 1. Fill in a length of 80. 2. Click on OK.	
15	Congratulations! Your first part is ready: an axis! A shape like this is called a 'Feature' in SolidWorks.	

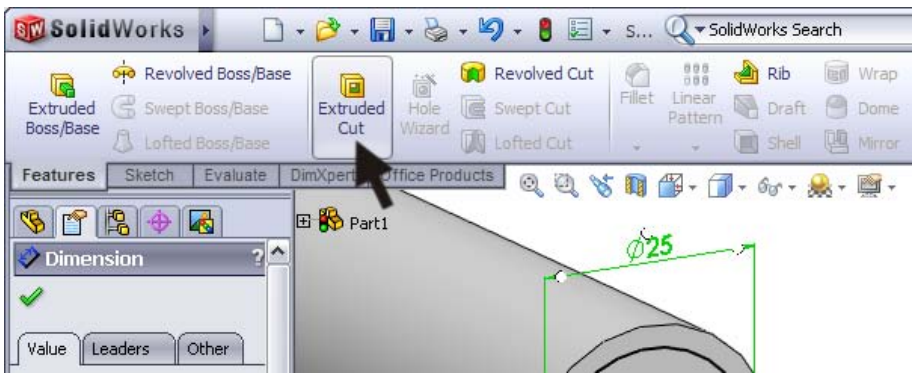
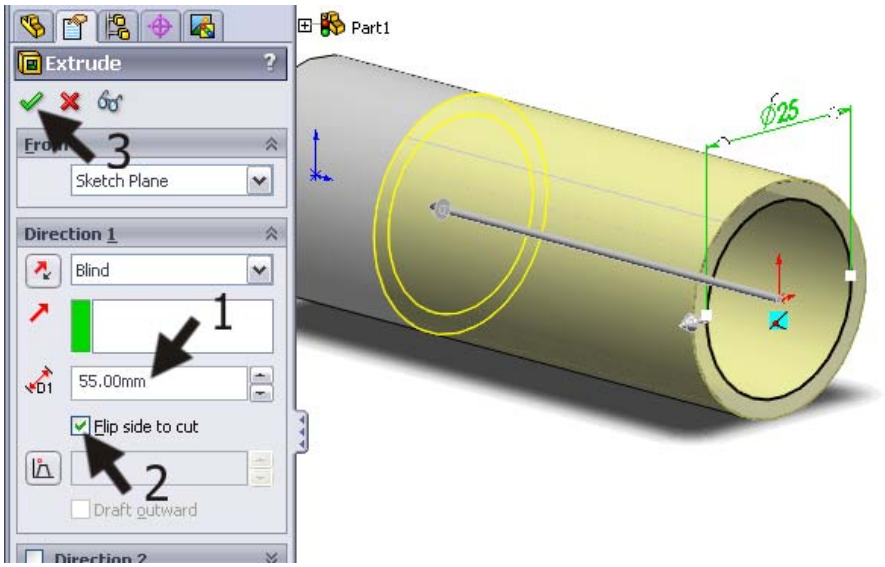
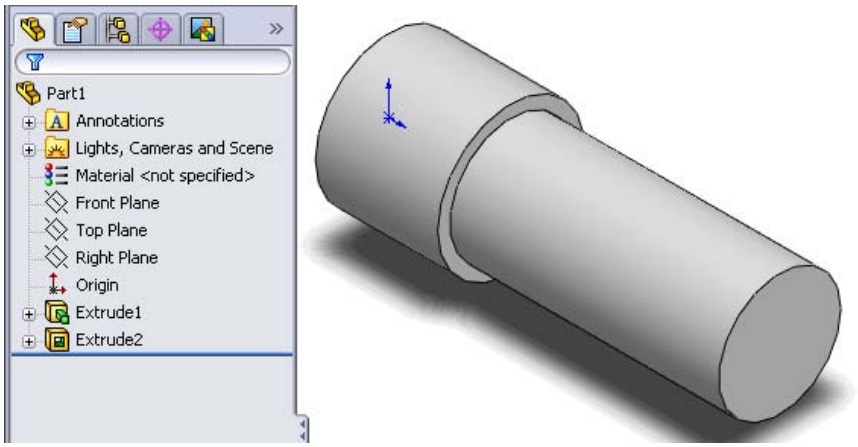
	Tip!	Sometimes the part you have created does not fit within the screen OR you may want to view it from another side. In SolidWorks, you only need the scroll-wheel from your mouse to change the view. • To zoom in or out: **turn** the scroll-wheel. The position of the cursor determines the position at which you are zooming. • To rotate your part: **push 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, just click on *View Orientation* at the top of the screen.  In the function menu that appears you can choose *Trimetric* to get the normal view back.
16	Next, we are going to make a new feature, but you need to make sure other actions have completely finished. Does the right upper corner of the screen look like the image on the right? This means the last action has not entirely finished. Click on the red cross to close the last command. Only then can you start a new one!	

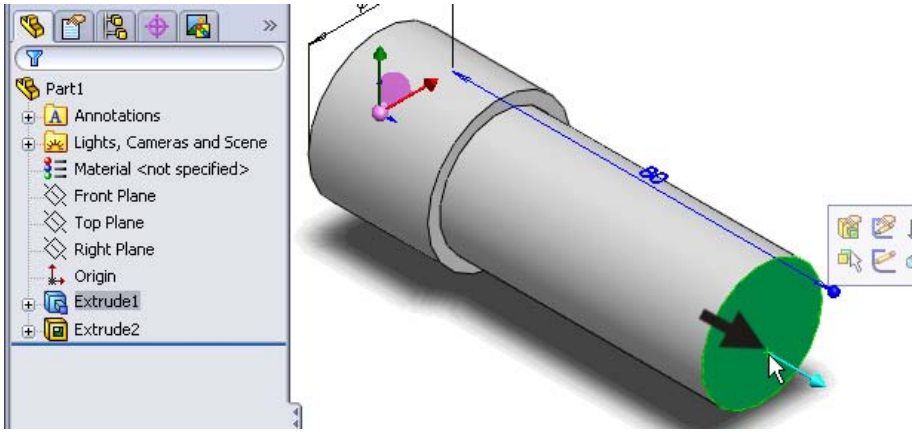
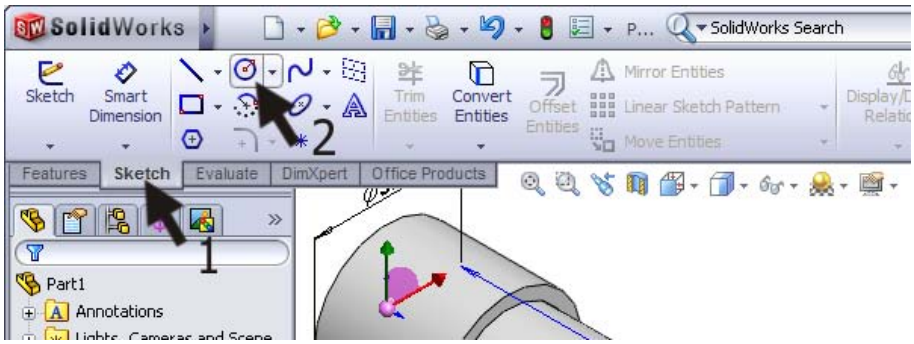
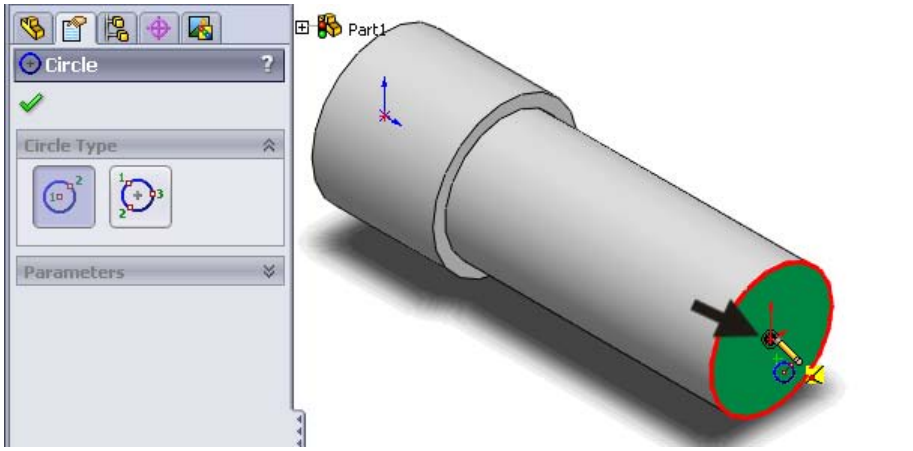
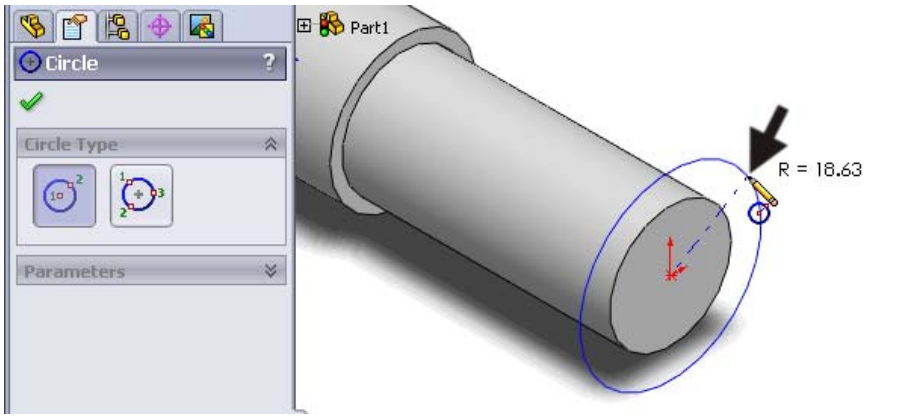
17	Next, we are going to change the diameter. Click on the top plane of the axis to select it. Be sure not to select the edge instead of the plane! When you do this right, the plane turns green.	
18	Click on Sketch to show the sketch commands.	
19	Click on Circle.	

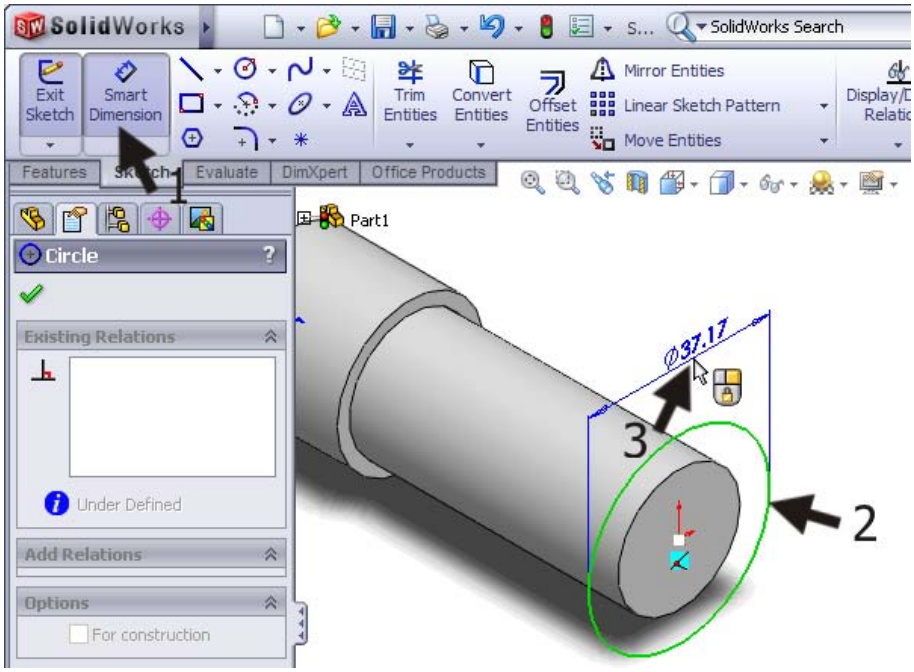
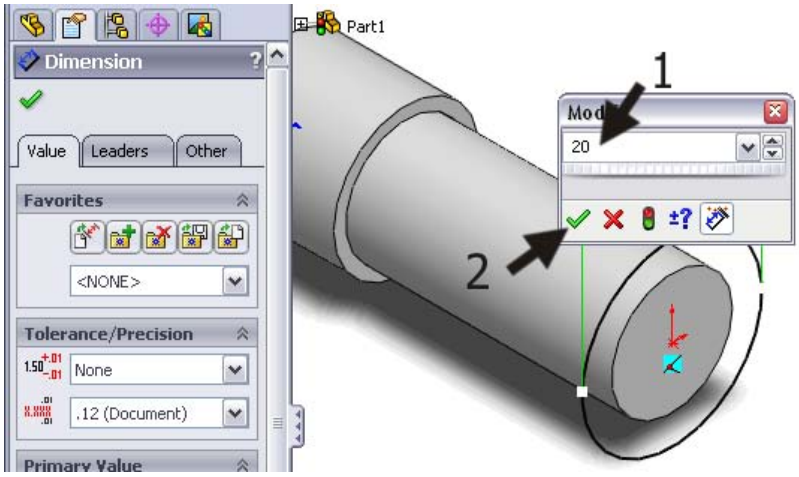
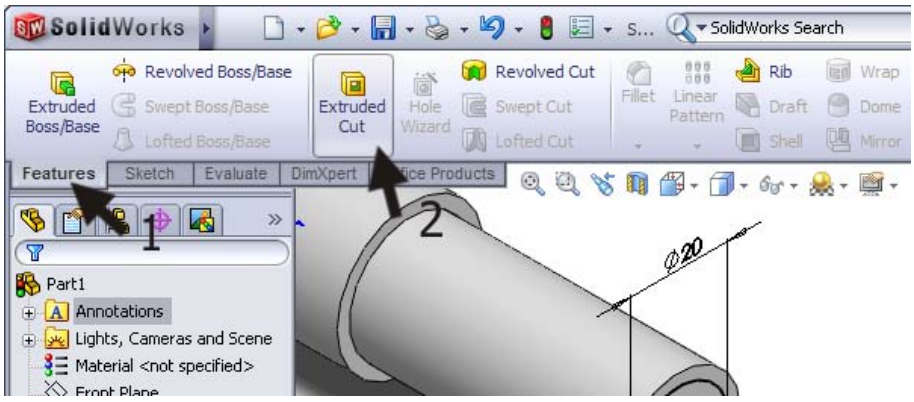
	Tip!	If you cannot get a clear view of what you are doing, zoom in or rotate your part. Remember: • To zoom in or out: **turn** the scroll-wheel. The position of the cursor determines the position at which you are zooming. • To rotate: **push the scroll-wheel** and move your mouse.
20	Point the cursor at the centre of the circle. The cursor changes like in the right image. Click only when the cursor has the right shape or you will fail to select the right item.	
	Tip!	Did you choose the wrong item or do you want to abort a command? Push the <Esc> key on your keyboard. You can also click the right mouse button and choose 'Select' in the menu that appears.  When you abort a command, you can start another one or throw away an element if you want. Click on the element in the sketch and push the <Del> (delete) key on your keyboard. (Pay attention: do NOT use the <Backspace>-button!).

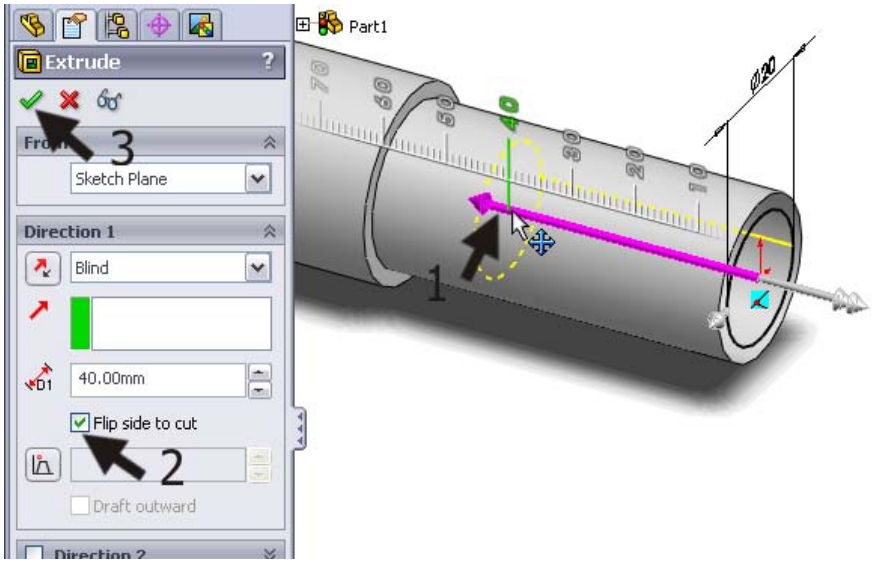
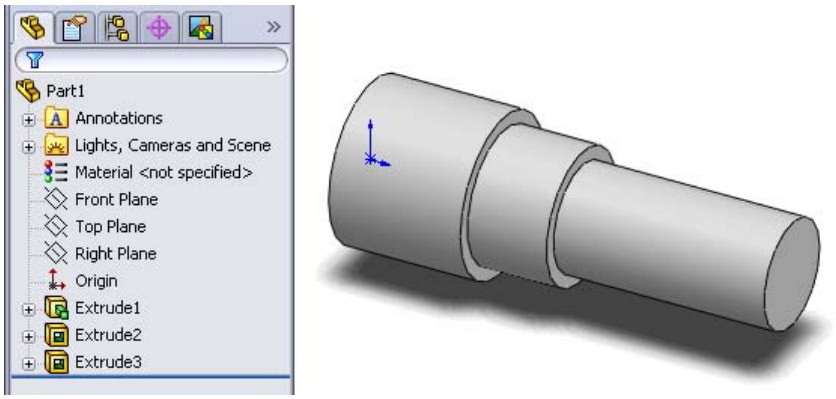
21	Move the cursor away from the center and click at any point to draw the circle. The dimension does not matter yet. **Pay attention: do NOT click on** another element like the outer circle of the plane.	
22	Click on 'Smart Dimension'.	
23	You have just drawn a circle. Next, click on it.	

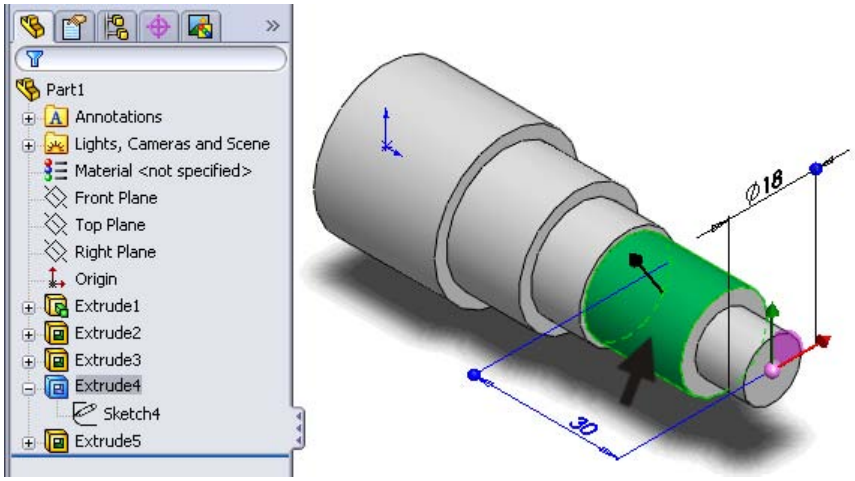
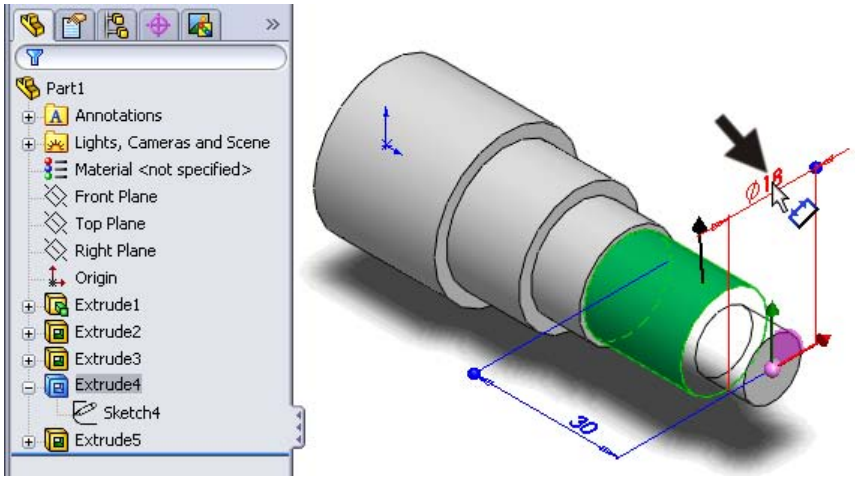
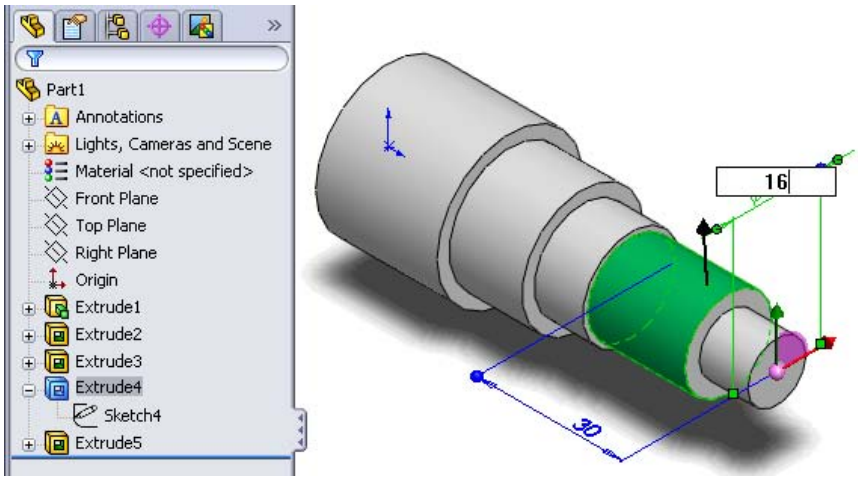
24	Move the cursor away from the circle and determine a position to enter the dimension. **Pay attention: do NOT click on** another element because SolidWorks will than calculate the distance between the circle and that element!	
25	A menu appears with which you can change the dimension. Change it to 25 and click on OK.	
26	Click on 'Features' to show the functions for adding or removing material.	

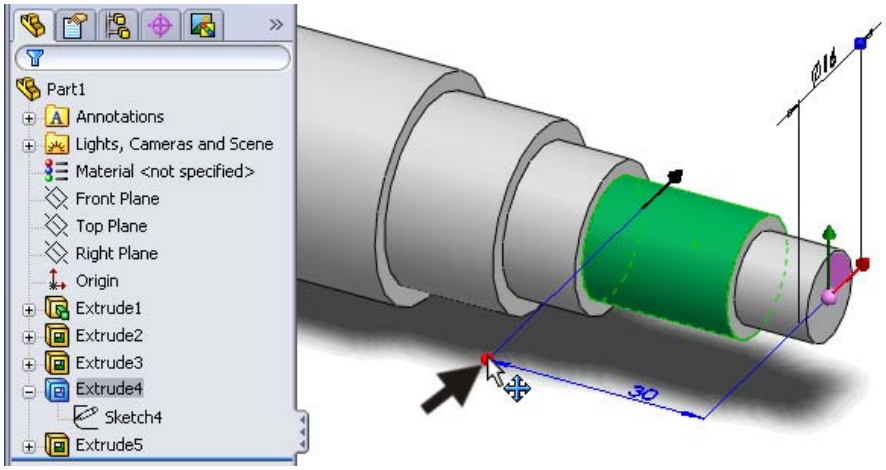
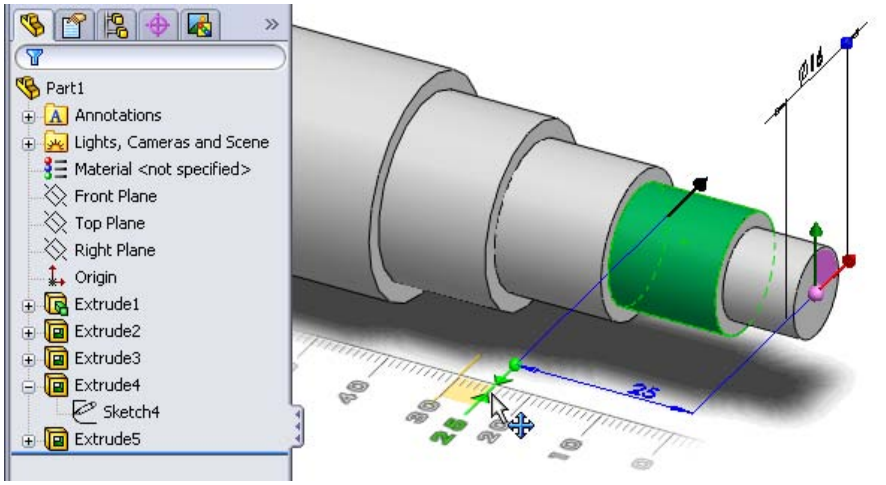
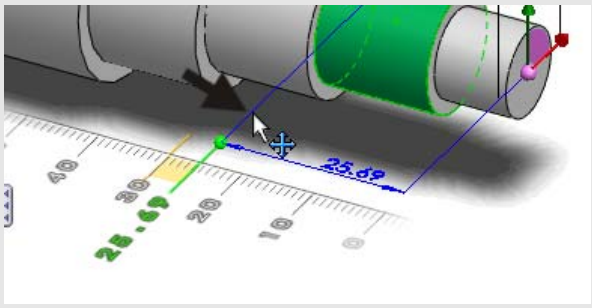
27	Click on Extruded Cut. You can remove material with this command.	
28	Next, enter the following features: 1. A depth of 55 2. Mark 'Flip side to cut' to make sure material on the **outside** of the circle, not the **inside,** is removed. 3. Click on OK.	
29	The first cut is made! We will make the second cut in exactly the same way. We will now speed up the steps to do so.	
30	Before making the next cut, make sure no command or sketch is active. Check the right upper corner. When a red cross like in the right image is visible, click on it to close the last command.	

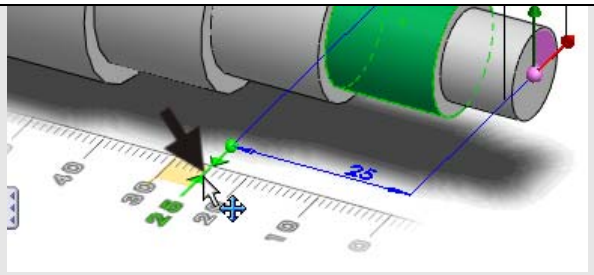
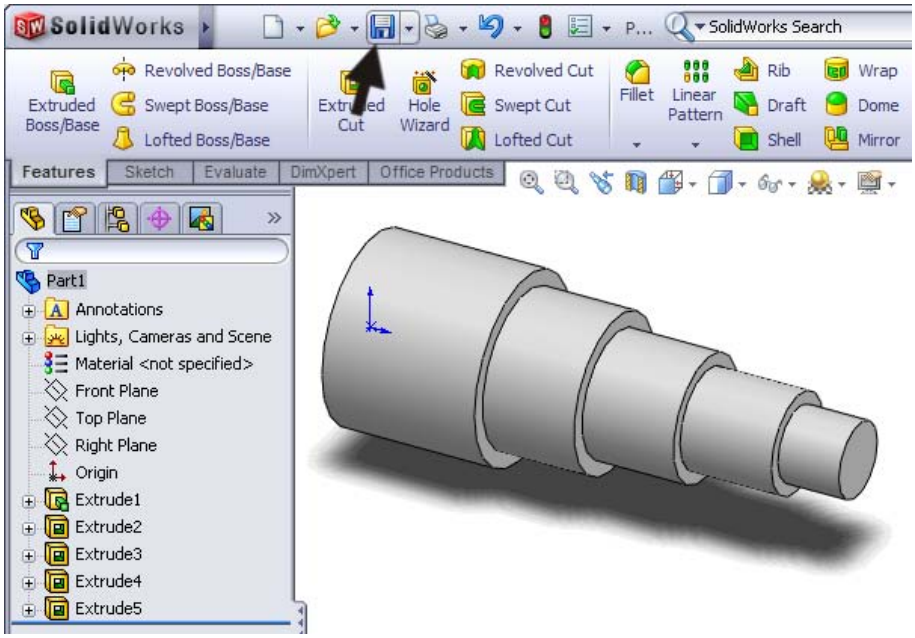
31	Select the end of the axis. Be sure to select the plane and not the edge!	
32	Click on Sketch first (to show the right functions) and then click on Circle.	
33	Click on the center of the axis. Notice the shape of the cursor!	
34	Click somewhere outside the material to draw a circle.	

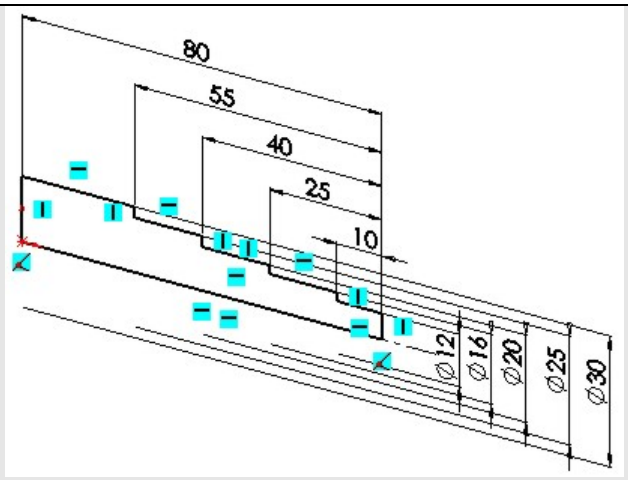
35	Next, enter a dimension for the circle: 1. Click on 'Smart Dimension'. 2. Click on the circle (it turns green, remember?) 3. Click above the part (**do not touch another element)** to position the dimension.	
36	Change the dimension to 20 and click OK.	
37	Click on 'Features' to show the right functions and next on Extruded Cut to remove material.	

38	Next, enter the following features: 1. Set the depth at 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!	At this point in the tutorial, you have learned two ways to set the depth of an extrusion: 1. You can enter the dimension in the field at the left of the screen, as you did in step 14 and 28. 2. You can drag the arrow in the part, as you did in the last step. Choose for yourself the way you think best.
39	The second cut is made!	
	Finish the part!	You need to make two other cuts in 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 you did before: 1. Check to make sure no command 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 notice that the dimensions of the third cut are wrong! It says Ø18x30, but it needs to be Ø16x25. How do we adjust this? In SolidWorks you will find this very easy to do! Click in the part on the third cut. The part dimensions will appear: Ø18 en 30.	
41	First, we adjust the dimension of Ø18. Click on this dimension once.	
42	Next, a small menu appears in which you can change the dimension. Enter 16 and push the <Enter> key on your keyboard. The part changes immediately to its new dimension.	

43	You can change the length of 30 in the same way, but we will now show you how you can also change this dimension 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**? If you are randomly dragging you will never get an exact dimension of 25 mm.  - Is the cursor pointing at the ruler? If so, 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 now completely finished! Click on 'Save' in the tool-bar and name the part axis.SLDPRT.	
	What are the most important items you have learned so far?	This first exercise is an introduction to SolidWorks. You have learned a few things that 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 start a new feature, be sure no other command is active and no sketch is still open. • You can easily adjust all dimensions. You will learn how to make more complicated adjustments, in one of the tutorials that follow.
	Is there another way to create this part?	Sure! You can create most parts with SolidWorks in several ways. There is no 'good' or a 'bad' way to do so. It's a matter of preference. In this exercise, we have created the part like you would on a turning machine in the workshop. This is often a good guideline for building a part. You could have also drawn the contour of the part and rotated it afterwards. In an exercise that follows, you will learn how to use this method in detail.


80
55
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Ø12
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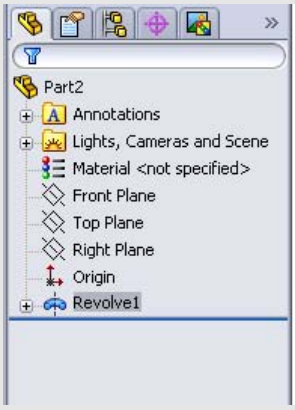
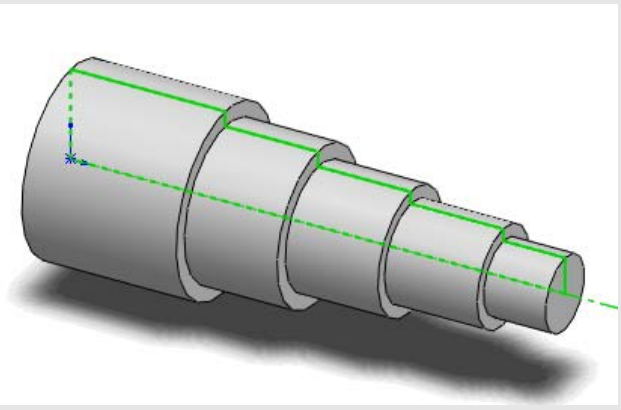

Part2
Annotations
Lights, Cameras and Scene
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Revolve1



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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 SolidWorks, 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

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

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