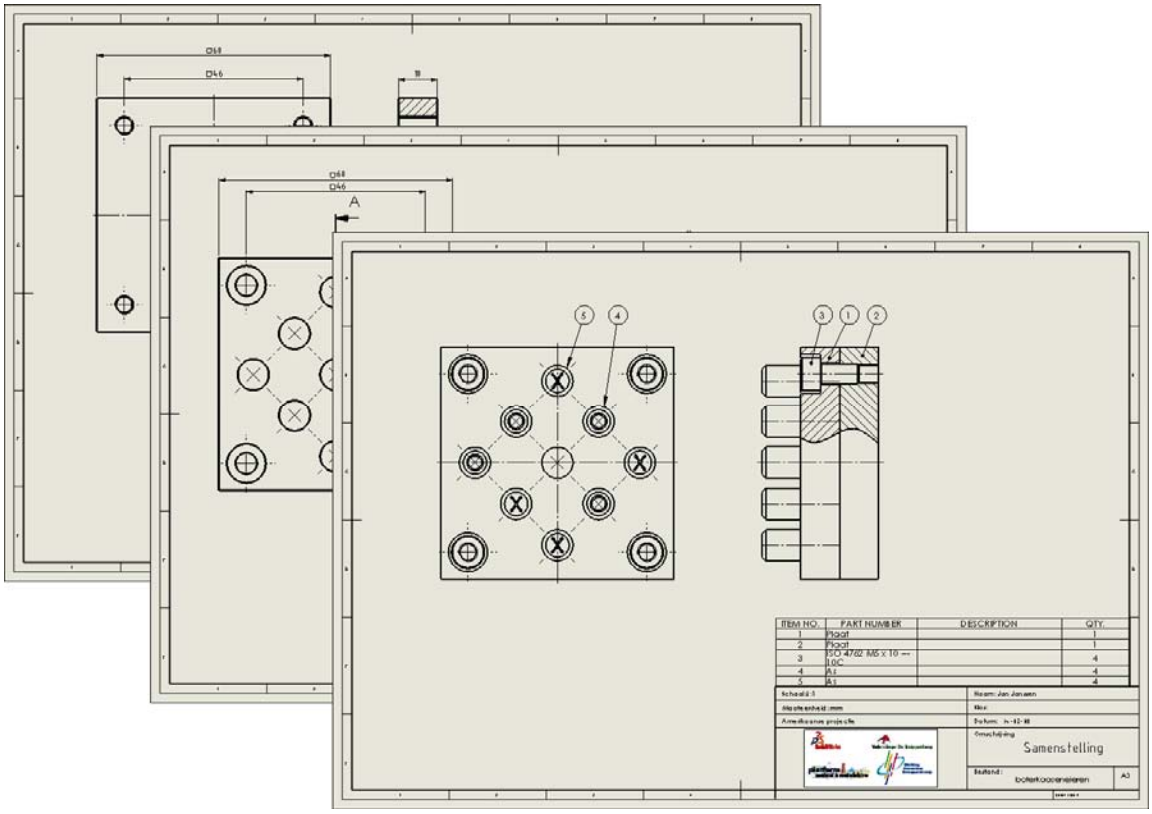
SolidWorks® Tutorial 6

DRAWINGS OF THE TIC-TAC-TOE GAME



Preparatory Vocational Training
and Advanced Vocational Training

To be used with SolidWorks® Educational Release 2008-2009

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Drawings of the TIC-TAC-TOE game.

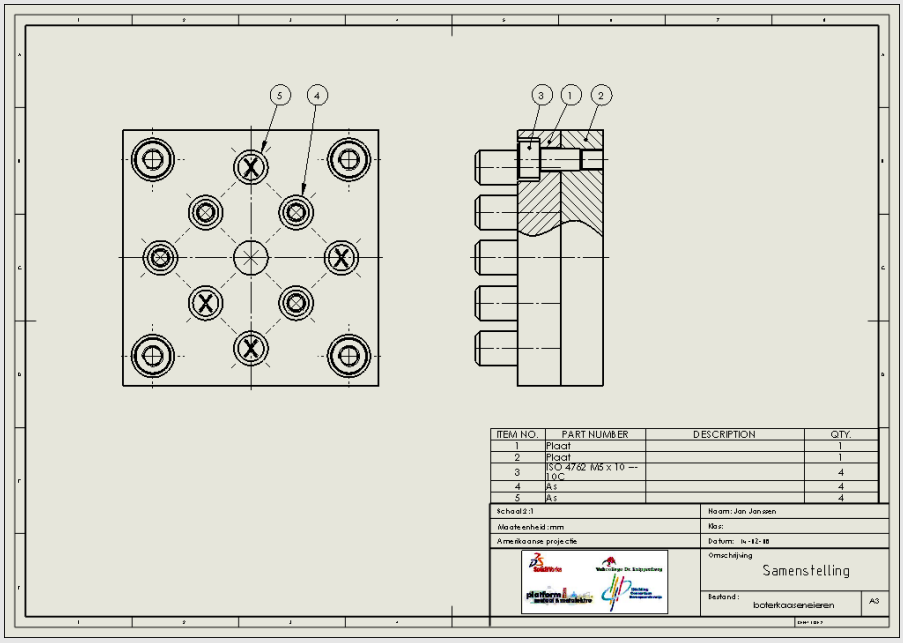
In this tutorial you will learn how to make a 2D drawing of a part that you have created in 3D. You must have completed Tutorial 5 first and saved the files associated with it in order to complete this tutorial.

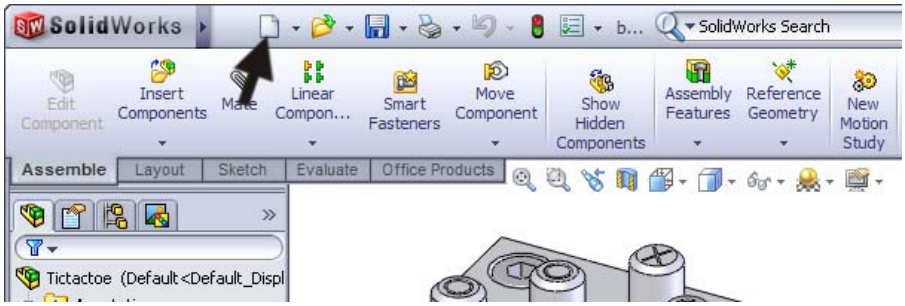
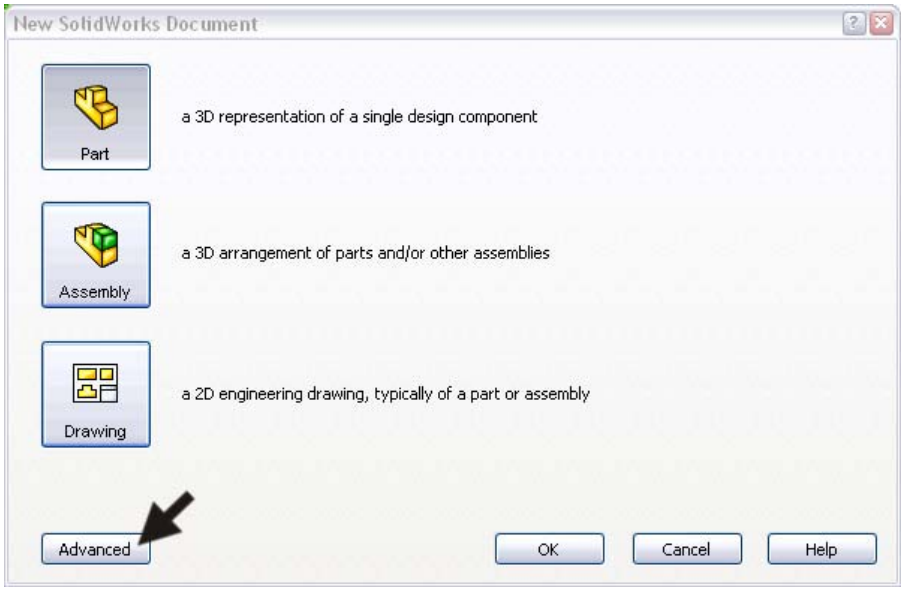
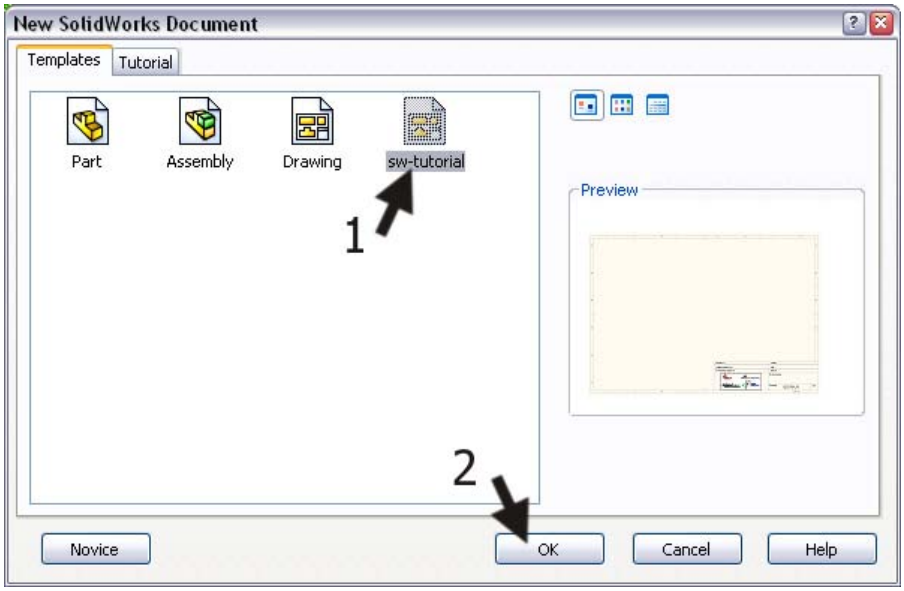
In this tutorial we will make the following drawings:

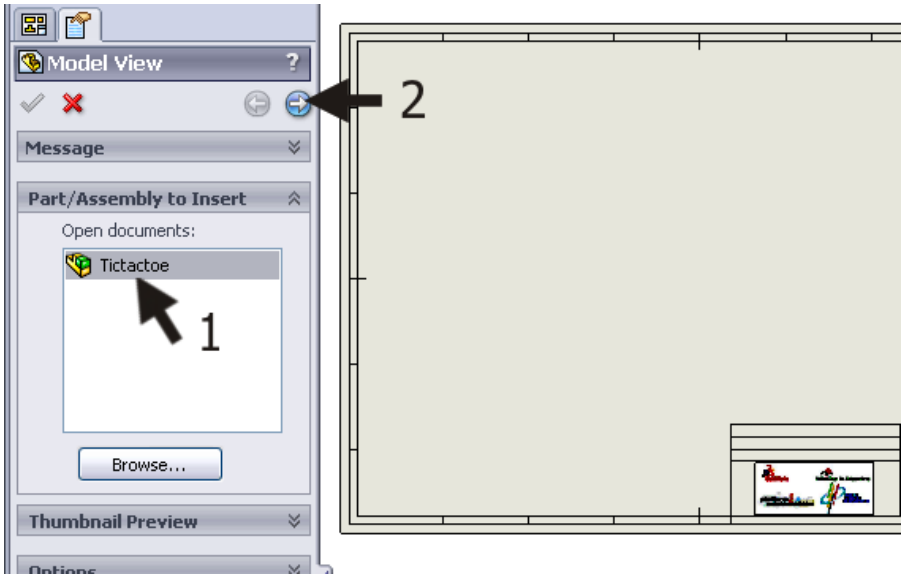
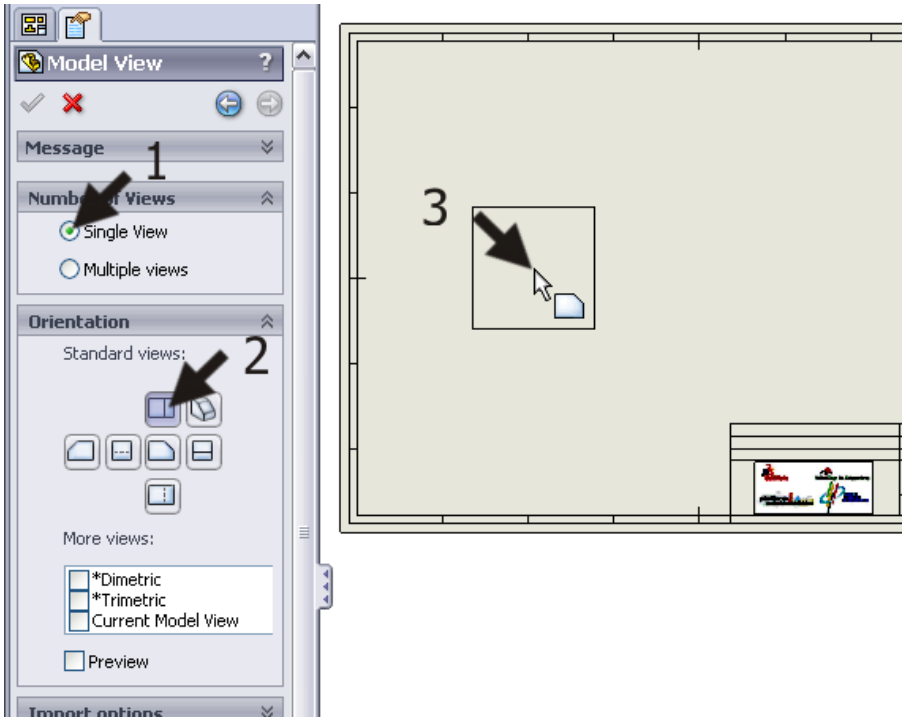
1. A drawing of the assembled parts.
2. A drawing of the bottom part, the base.
3. A drawing of the top part.

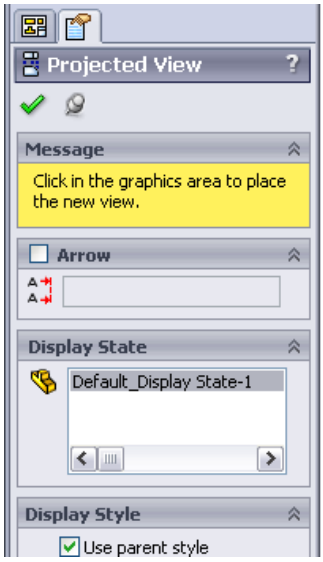
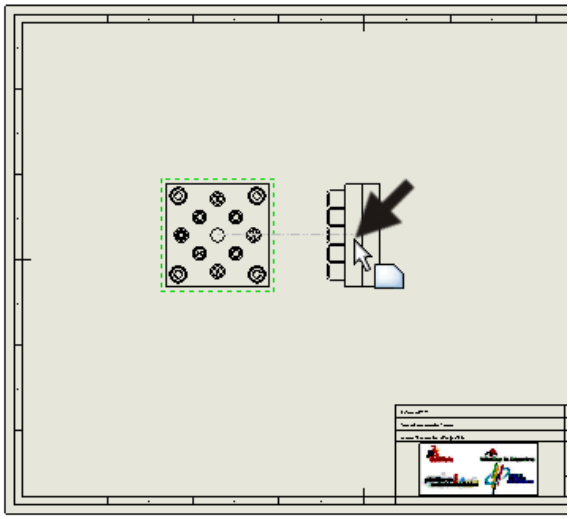
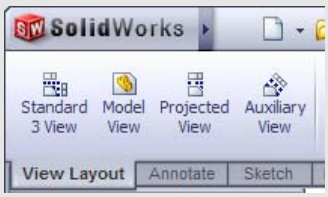
Work plan

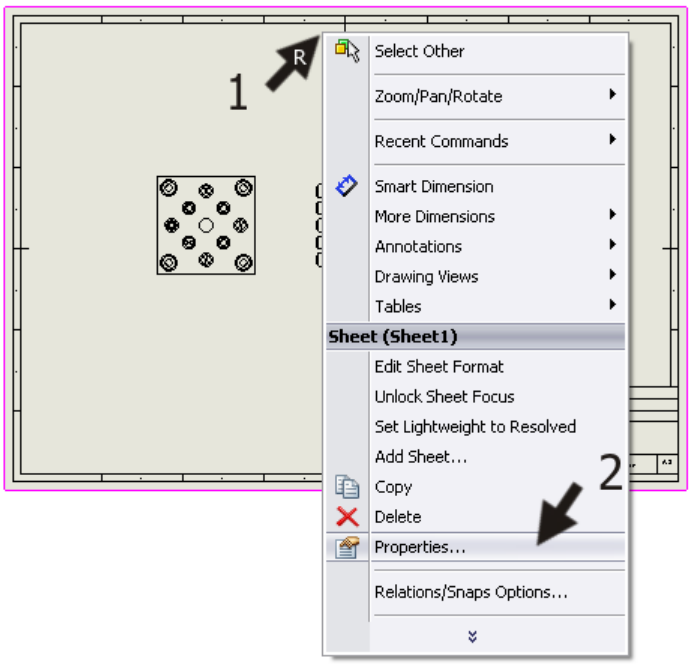
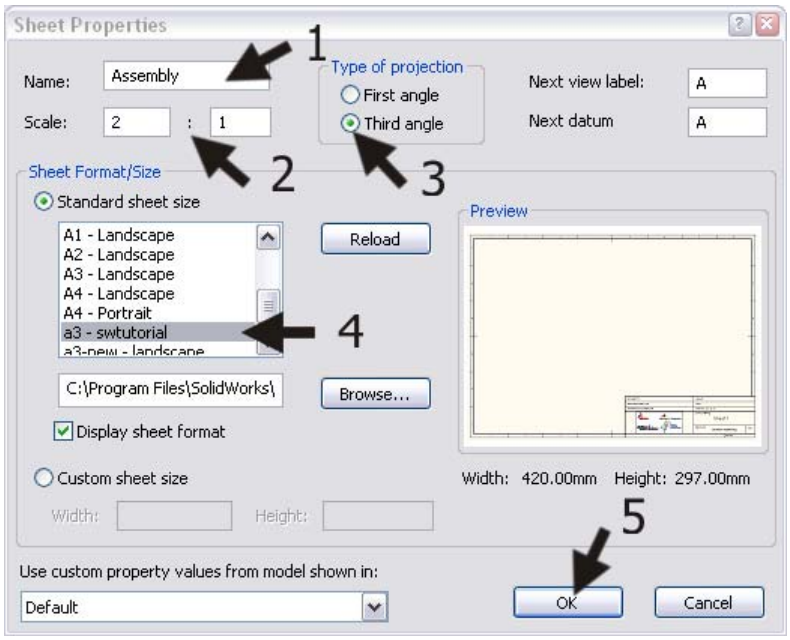
First, we will make an assembly drawing. We will use the top and side views with a partly transparent side.

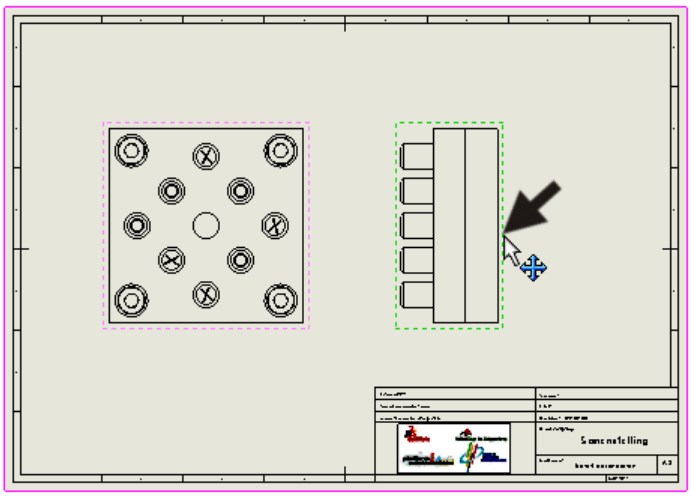
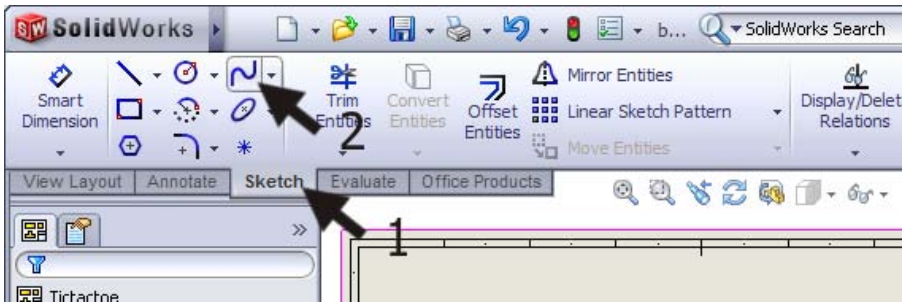
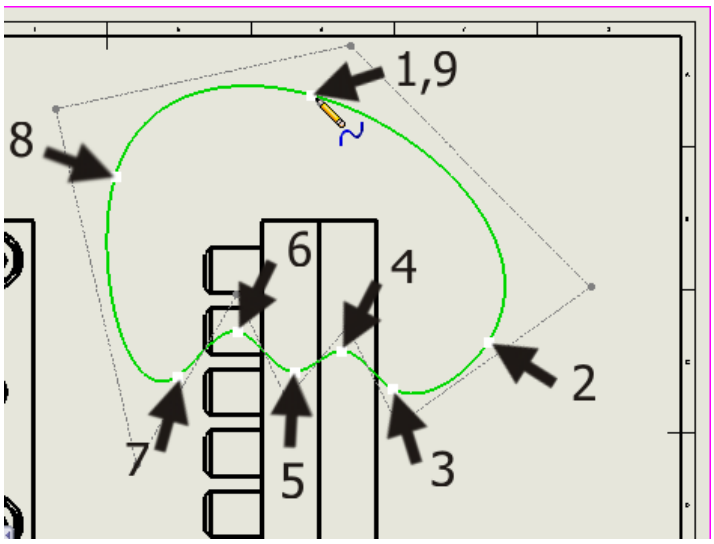


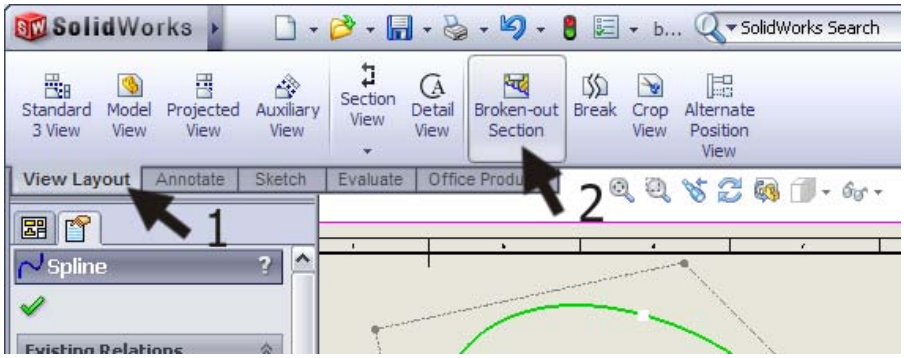
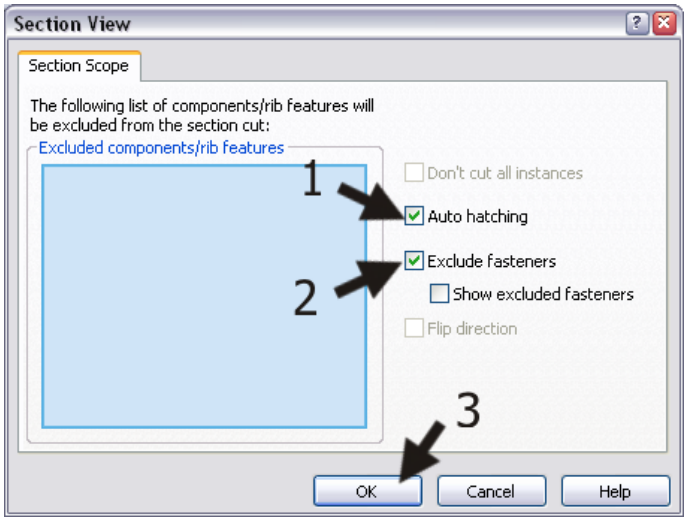
1	Start SolidWorks and open the assembly Tictac-toe.SLDASM, which you have made in the last tutorial.	
2	Click on New in the Toolbar.	
3	Click on 'Advanced' in the menu that appears.	
4	1. Select the template 'sw-tutorial' (SolidWorks Tutorial). 2. Click on OK. Whenever this template is not available, ask your teacher about it. Do you work at home? If so, you can download the file templates.zip from www.solidworks.nl. An explanation about where to put your files is included in the ZIP file.	

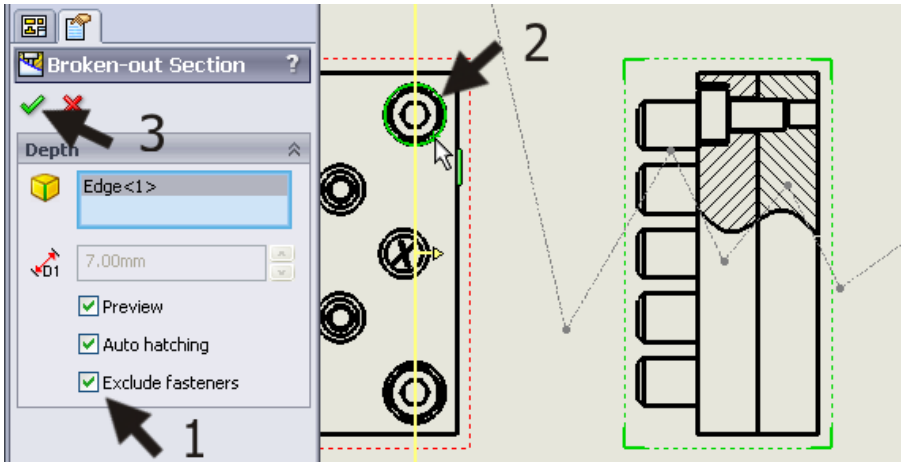
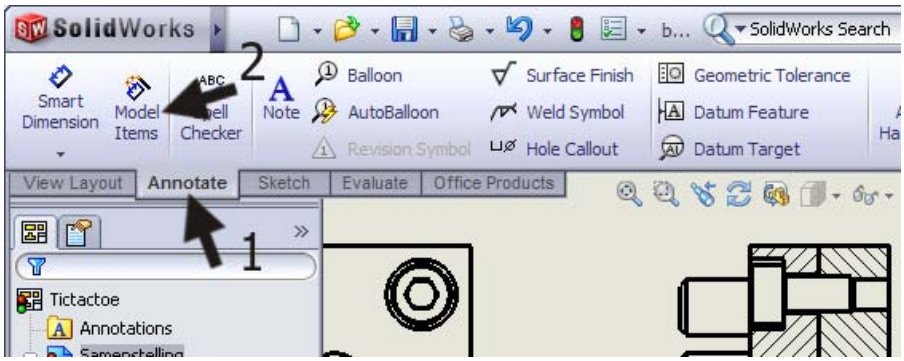
5	1. Select the file 'Tictac-toe'. 2. Click on 'Next'.	
6	1. Select 'Single View' in the PropertyManager (to place ONE view in the drawing). 2. Select the Top View. 3. Position the view on the drawing board.	

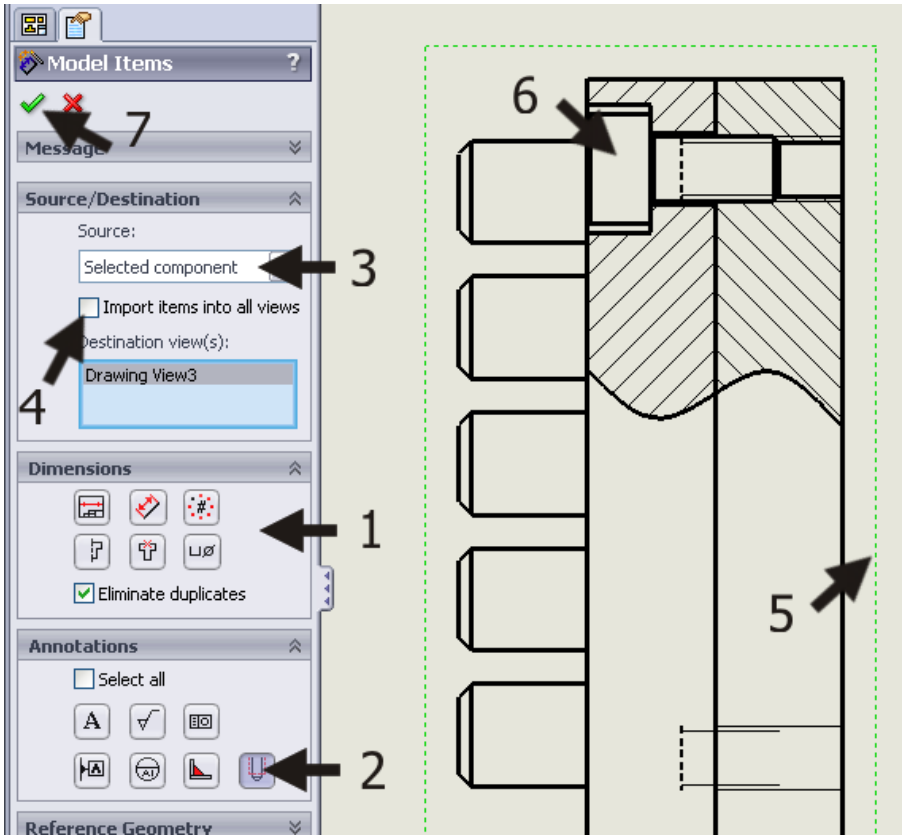
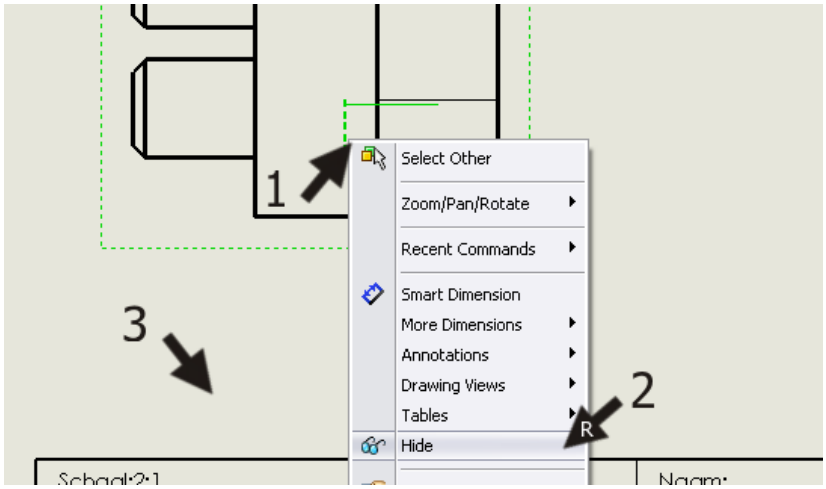
7	After you have positioned the view, SolidWorks will automatically start the command 'Projected View'. Click beside the top view to put a side view next to it. Push the <Esc> key on your keyboard to end this command.	 
	Tip!	There are three commands for placing views on your drawing board: **Model View**: this is used to place one of the main views in the drawing field. This is actually the same method you used in steps 4 and 5. **Projected View**: with this command you can extract a view using the American or European projection method from the existing file. **Auxiliary View**: this command is used to extract an auxiliary view from the existing view and place it at a random angle to the main view.  With **'Standard 3 View'** you will select the three main views (Top, Front, and Right) with only one mouse click and place them on your drawing board.

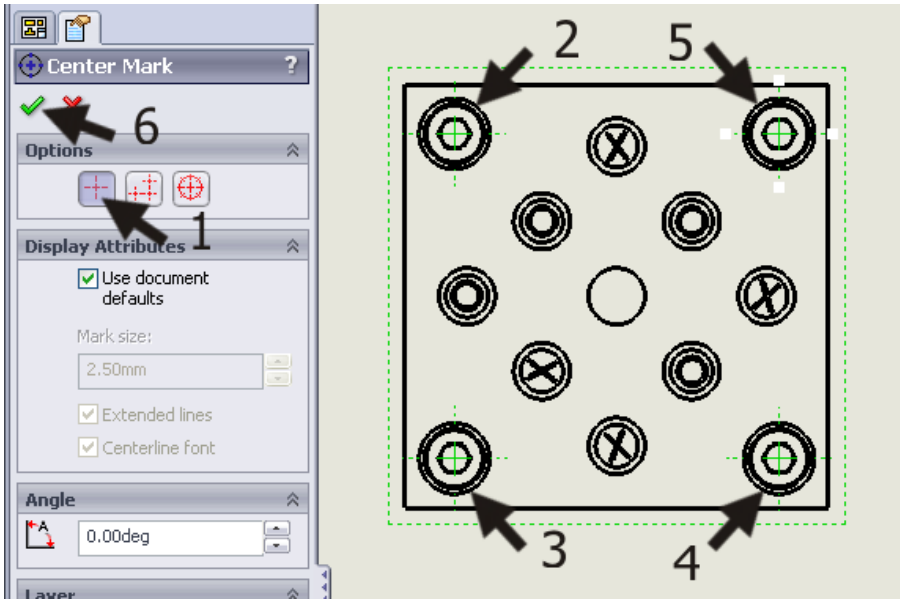
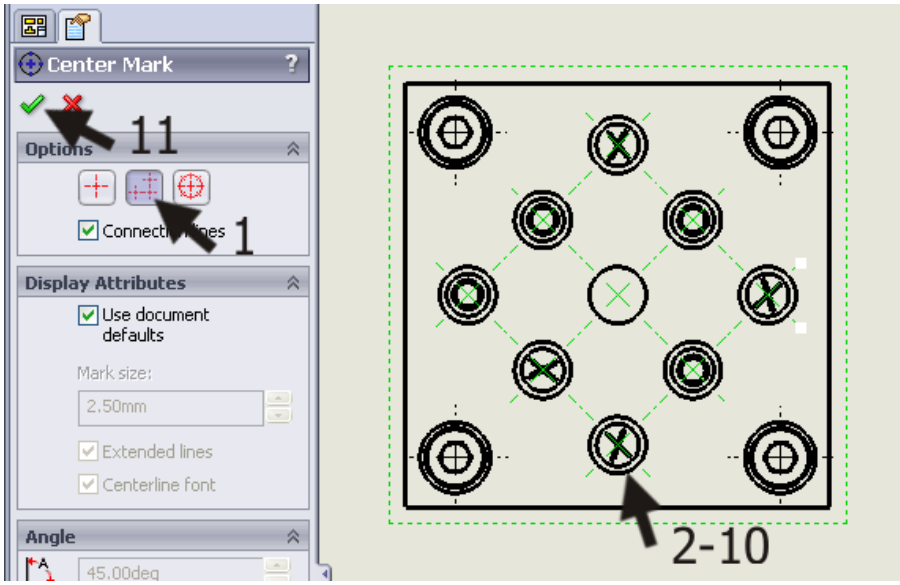
8	1. Right-click at a random position somewhere on the drawing board (not on a view!). 2. Select: 'Properties' in the menu that appears.	
9	1. Name the drawing: 'Assembly'. 2. Set the scale to '2:1' in the menu that appears. 3. Select 'Third angle' for 'Type of projection': 4. Select the paper size 'a3 – swtutorial': 5. Click on OK.	
	Tip!	In the Netherlands, the American projection is used for all technical drawings and designs. This is called Third Angle Projection. In most other European countries, the European projection method is used. This is called First Angle Projection. We will be using the Third Angle Projection, but of course you can choose to use the First Angle Projection. The views will relate to on another in a different way.

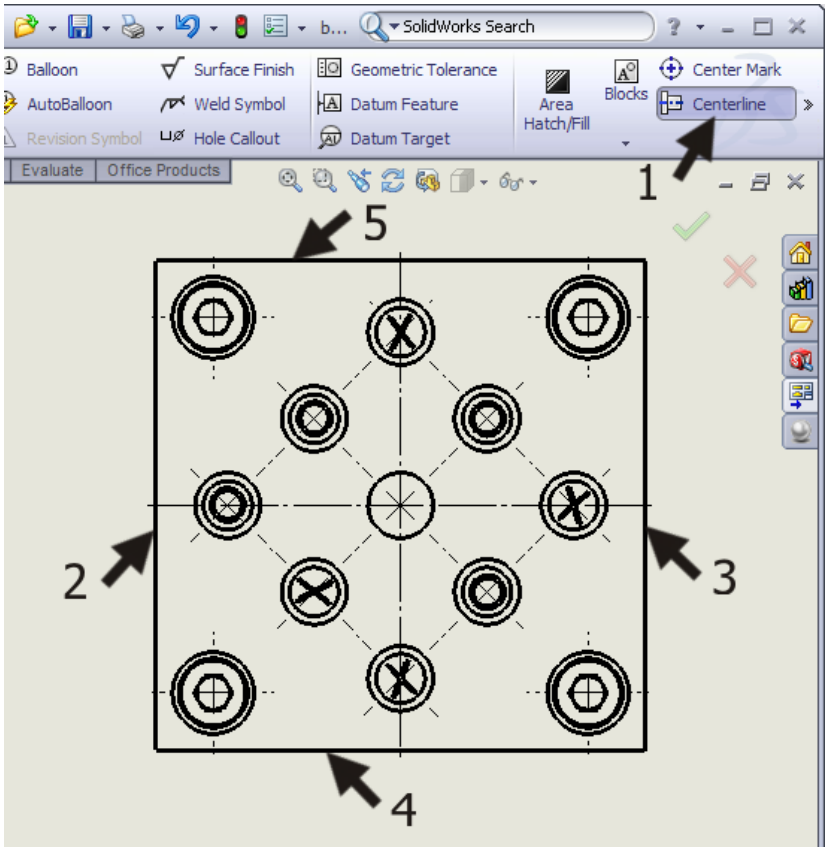
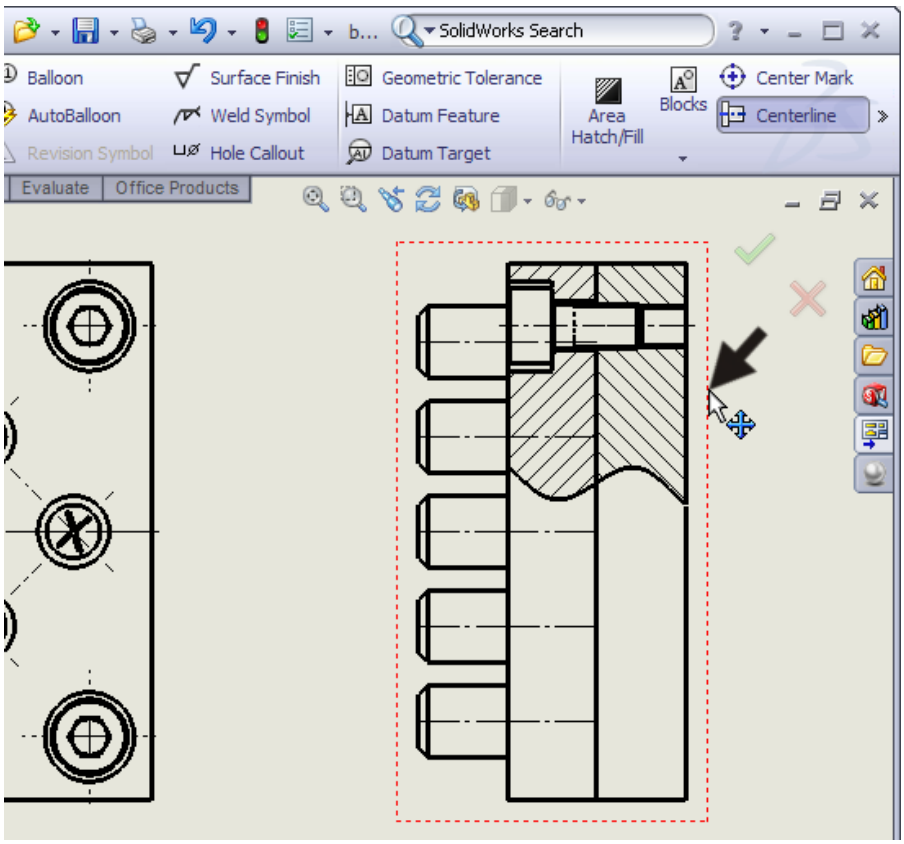
10	When you move your cursor over a view, a dotted frame appears around the view. With this frame, you can drag the view to adapt the way the views are positioned on the drawing board. Be sure the views are neatly aligned in the middle of the drawing board.	
11	Next we a portion of the side view transparent to provide a clear view of the hexagonal bolt. 1. Click on 'Sketch' in the CommandManager. 2. Click on Spline.	
12	Draw a curve as shown in the illustration on the right. You will position several random points in the drawing. Try to copy the shape as shown on the right. Be sure the last point is in the same position as the first one. Only then will you get a closed curve.	

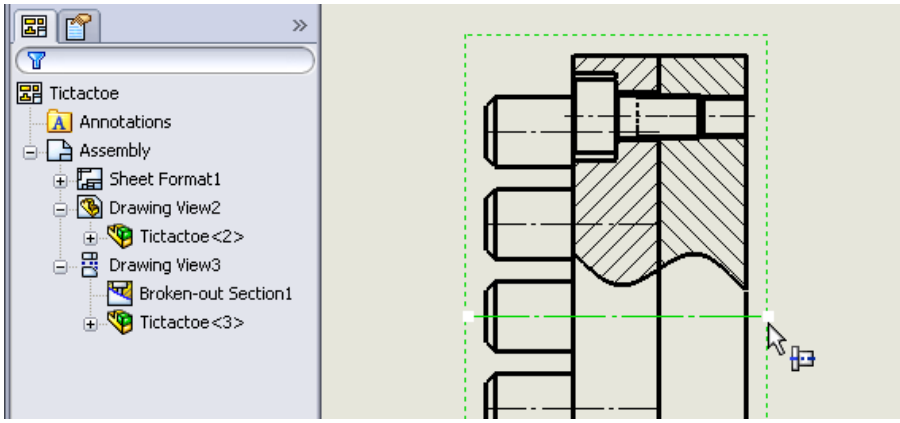
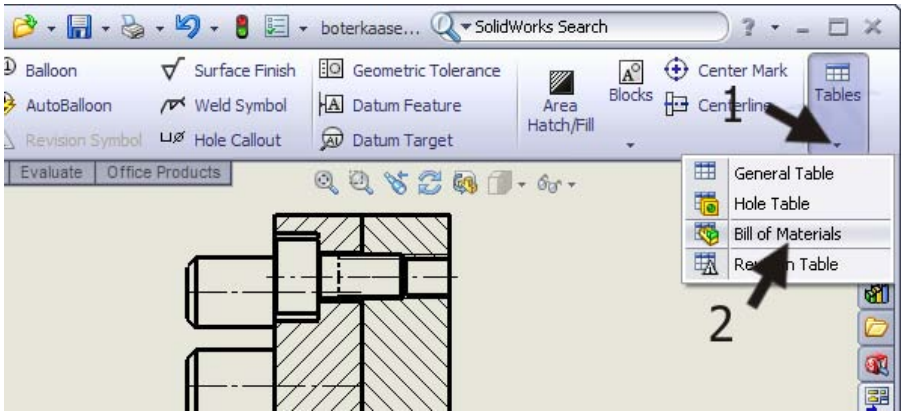
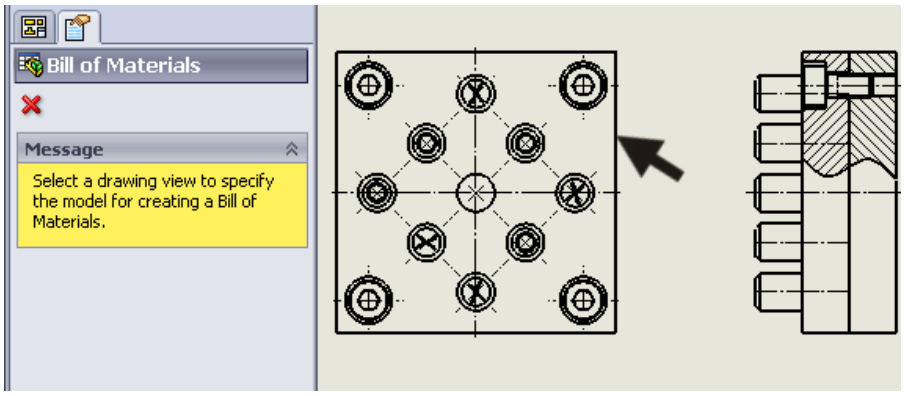
13	Be sure the curve you have just drawn is still selected (green). 1. Click on 'View Layout' in the CommandManager. 2. Click on 'Broken-out Section'.	
14	Next, set the features in the menu that appears: 1. Check 'Auto hatching'. 2. Check 'Exclude fasteners'. 3. Click on OK.	
	Tip!	The menu you have seen in step 14 will always appear when you have made a broken-out section from an assembly like we just did. You can set a few items in this menu: **Auto hatching**: this option makes sure that different parts are hatched in different directions. When you fail to check this option, hatching occurs without differences through all parts. **Excluded components:** in the blue field, you can select parts to break out. **Exclude fasteners**: fasteners, like the hexagonal bolts in our drawing, stay complete.

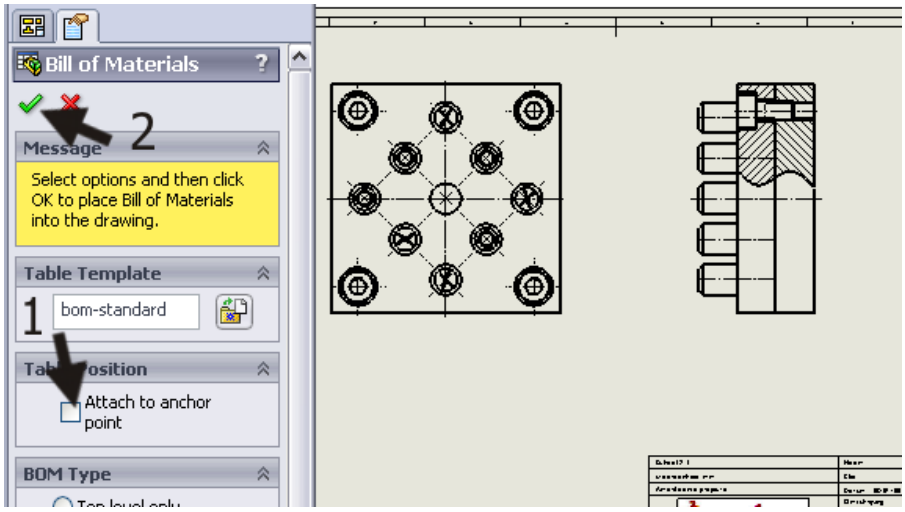
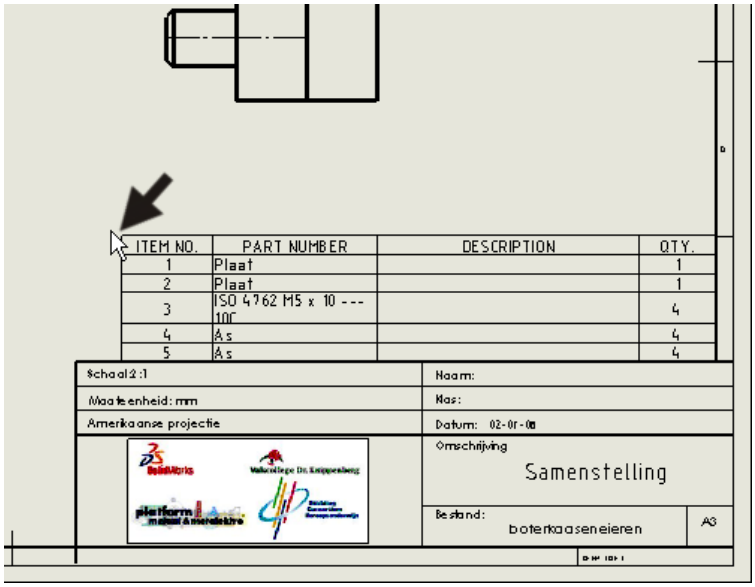
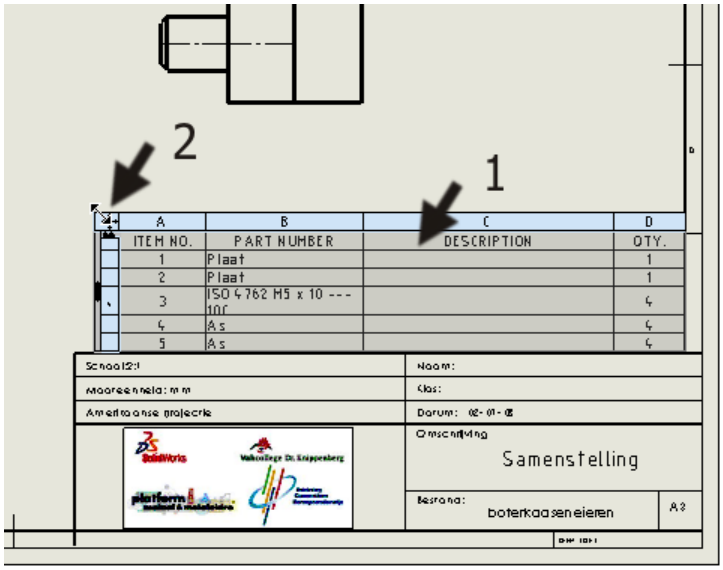
15	1. Be sure that all three options at the bottom are checked ('Preview', 'Auto hatching' and 'Exclude fasteners'). 2. Next click on the hole of the hexagonal bolt. In this way, you determine the depth of the break-out. The yellow line now goes through the middle of the circle. 3. If the preview looks all right, click on OK to finish it.	
16	As you can now see, the thread of the hexagonal bolt and the base plate are not shown. In an assembly you must do as following: 1. Click on 'Annotate' in the CommandManager. 2. Click on 'Model Items'.	

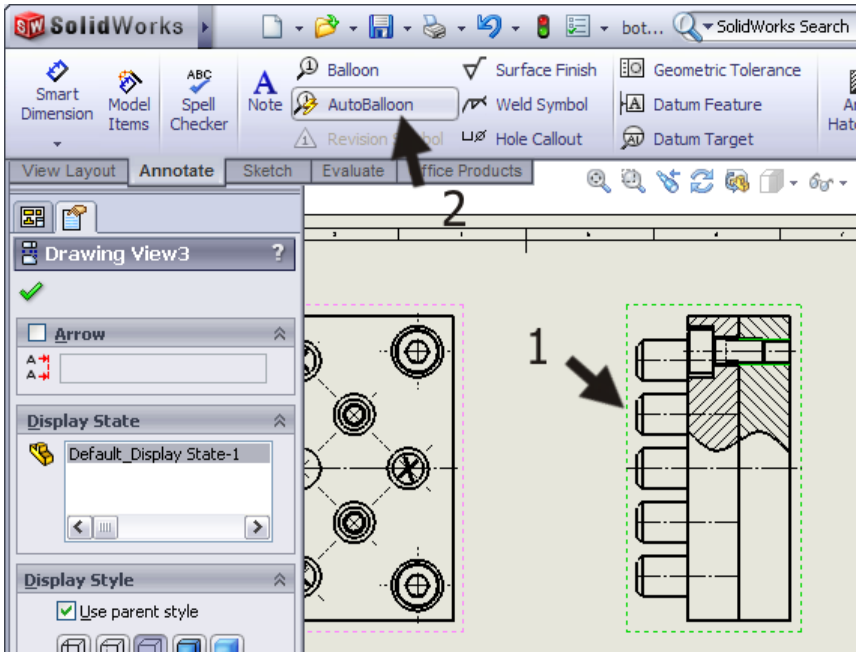
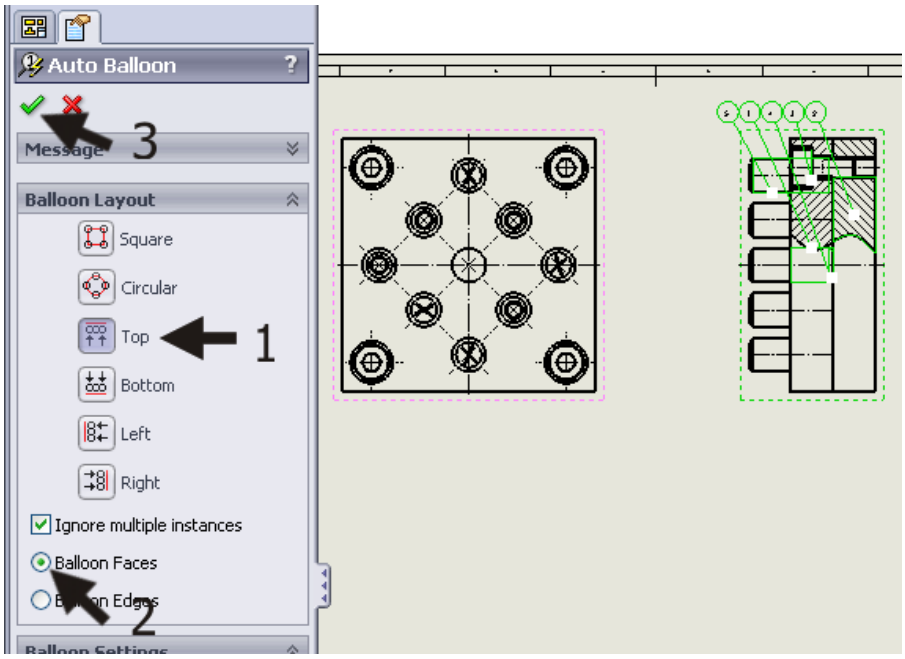
17	Set the next features in the PropertyManager: 1. Be sure to set all 'Dimensions' buttons OFF. 2. Check the Cosmetic Thread in the 'Annotations' field. 3. Select 'Selected component' in the 'Source/Destination' field. 4. Uncheck the option 'Import items into all views'. 5. Click on the frame of the view in the drawing. 6. Click on the drawing of the hexagonal bolt. The thread features are added at this point. 7. Click on OK.	
18	As you can see, the thread is also revealed at the bottom hexagonal bolt (which should not be visible. We have to hide it: 1. Right-click on the thread. 2. Click on 'Hide' in the menu that appears. 3. Click beside the view to check if the thread turned invisible. The thread is still visible, because there are TWO holes directly on top of each other. Therefore, repeat steps 1 to 3. Do the same for the thread in the base plate.	
19	Next, we are going to place the centerlines in the top view. Click on 'Center Mark' in the CommandManager.	

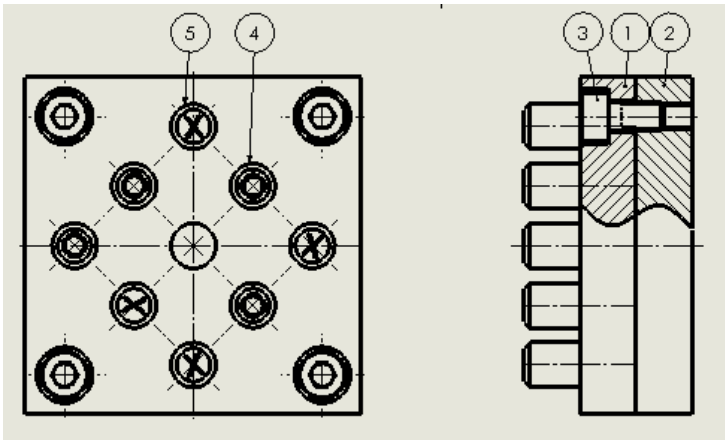
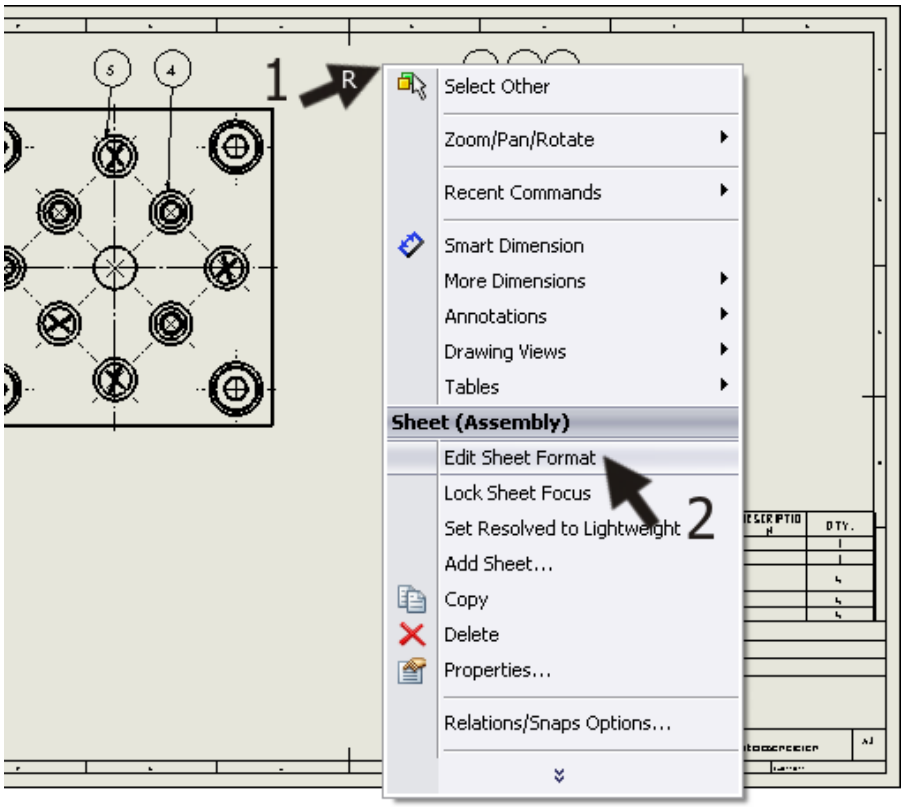
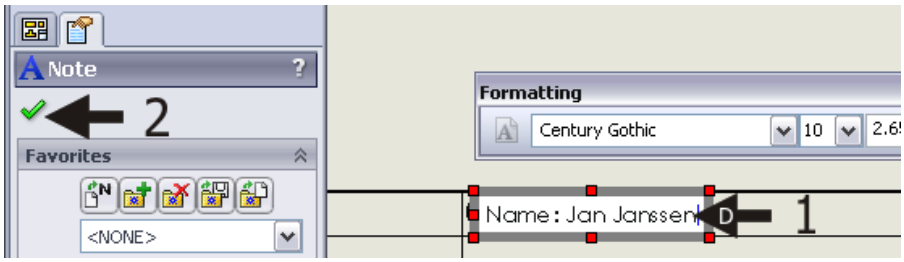
20	1. Be sure the first button (Single Center Mark) in the PropertyManager is checked in the 'Options' field. 2-5. Click on the four holes at the outer ends of the base plate. 6. Click on OK.	
21	Select the command 'Center Mark' in the CommandManager again. (Look at step 19). Set the following features in the PropertyManager: 1. Click on the second button in the 'Options' field. (Linear Center Mark). 2-10. Click on the outer circles of all nine cylinders. 11. Click on OK.	

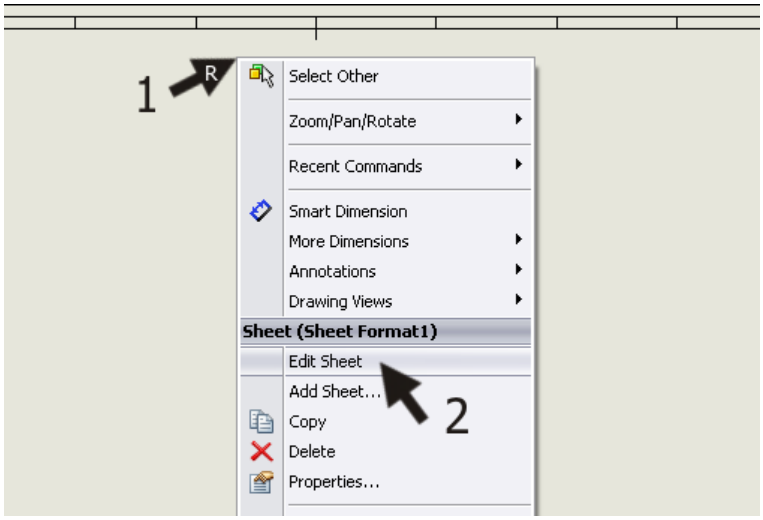
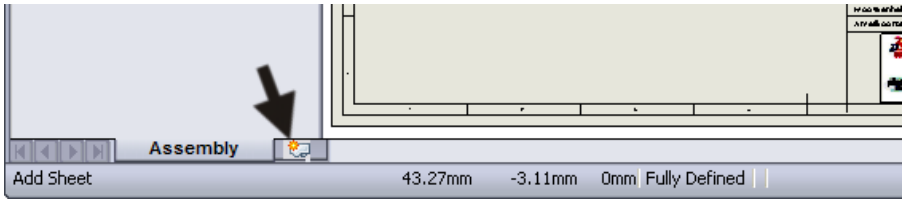
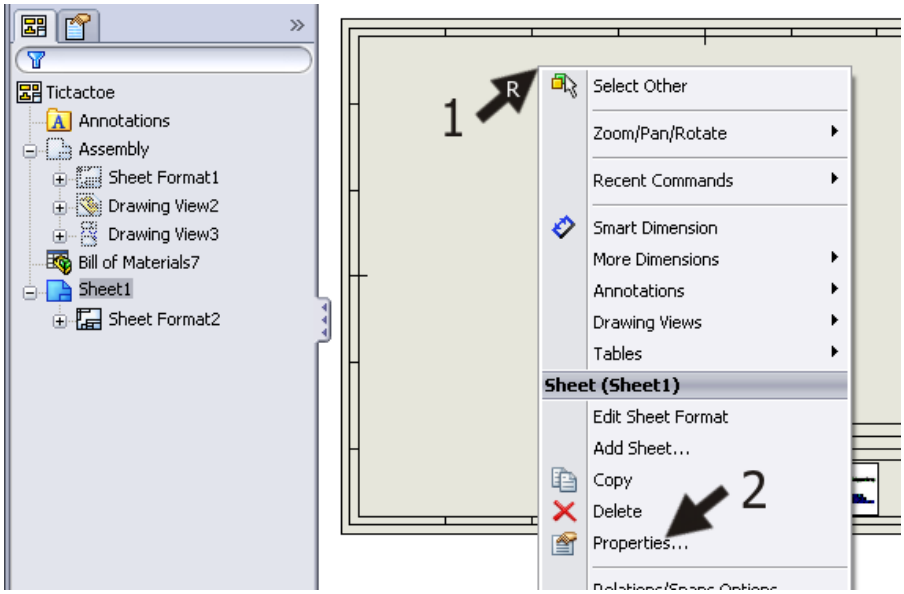
22	1. Select the command 'Centerline' in the CommandManager. 2,3 Next, click on the two vertical sides of the square. The vertical centerline is placed in the view. 4,5 Next, click on the two horizontal sides to place a centerline.	
23	Next, we draw the centerlines in the side view. Click on the command 'Centerline' again (look at step 22). Click on the frame which is around the view. All centerlines are automatically placed now. Pay attention: if this does not work, close the command and try again!	
	Tip!	In step 23 we have placed all centerlines in a single action. This is very

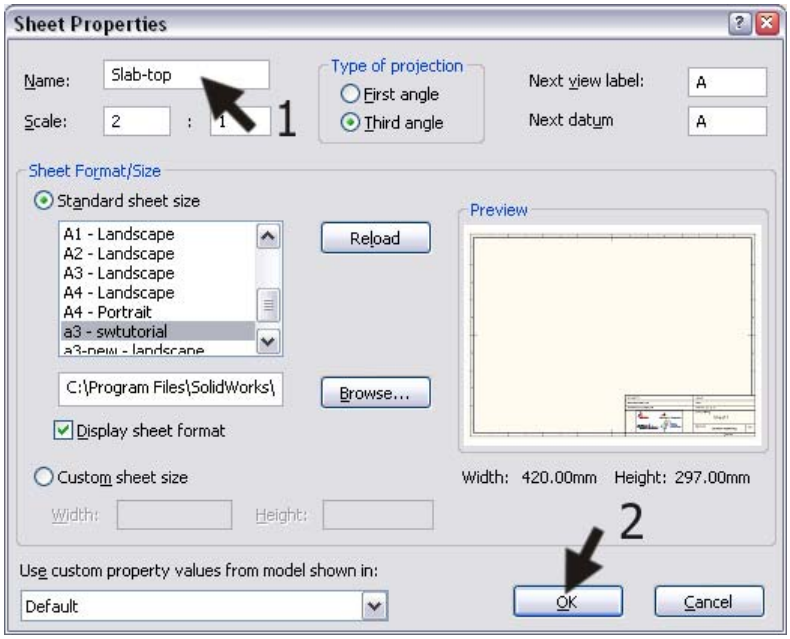
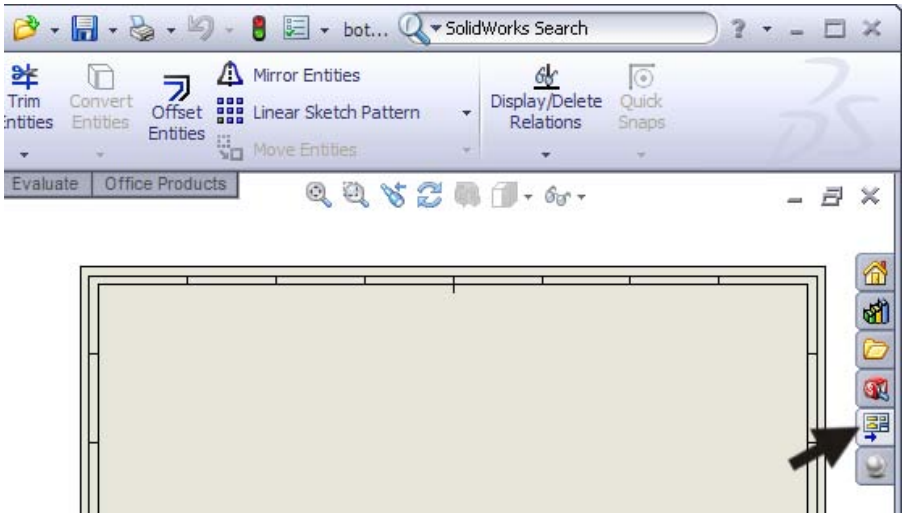
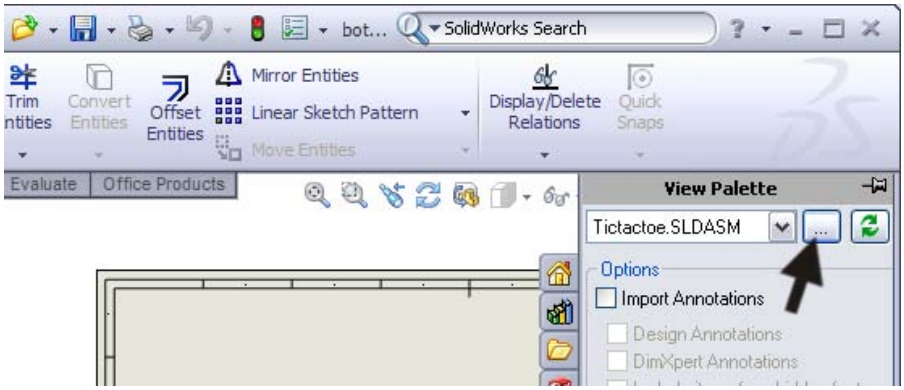
		convenient of course, but sometimes we will get more centerlines then we need. If this is the case, you can simply delete with the <Del> (delete) key on your keyboard.
24	Now, we want to extend the centerline that is in the middle. Click on the center-line and drag the ends a bit, as shown in the illu-stration.	
25	Next, we will put a parts list on the drawing board. It is called a Bill of Mate-rials. 1. Click on 'Tables' in the CommandManager. 2. Click on 'Bill of Mate-rials'.	
26	Click on one of the views.	

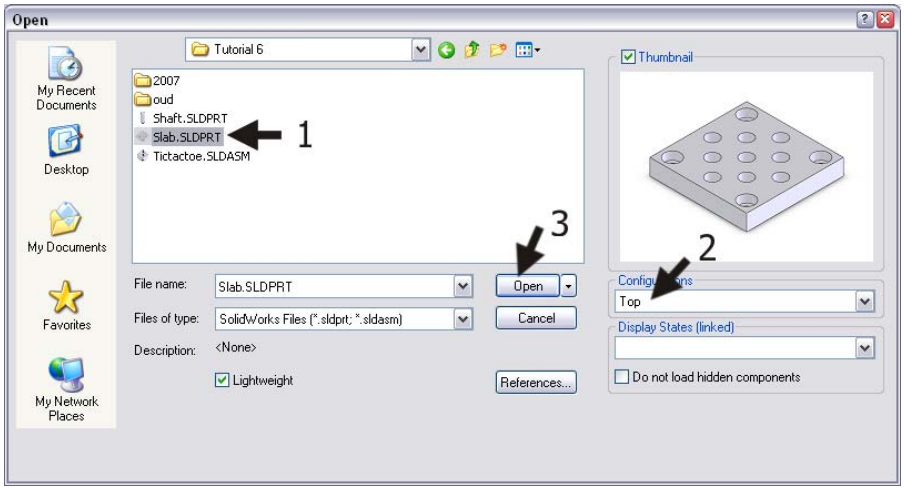
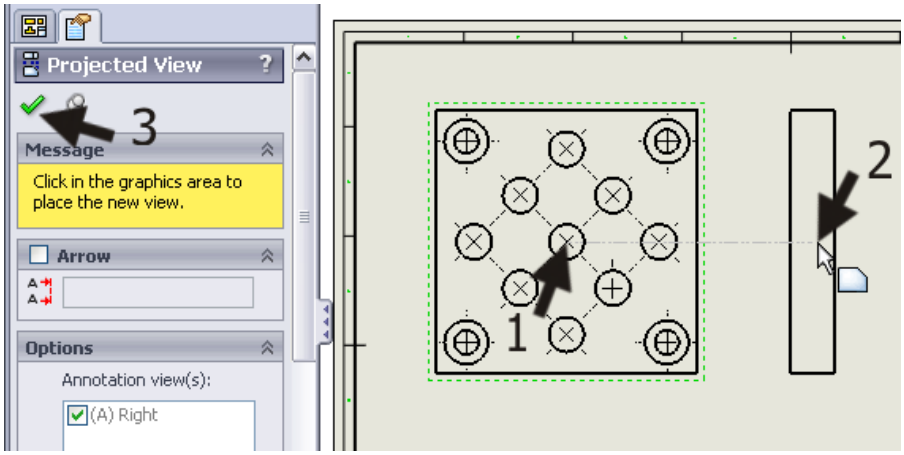
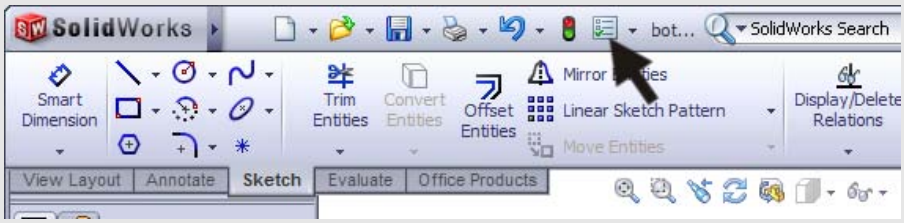
27	1. Uncheck the option 'Attach to anchor point' in the PropertyManager. 2. Click on OK.	
28	Place the parts list just above the title block of the drawing.	
29	To adapt the size of the parts list, do the following: 1. Click somewhere in the parts list to select it. Blue bars will appear on the left and right. 2. Drag the left top corner from the parts list to the desired position.	

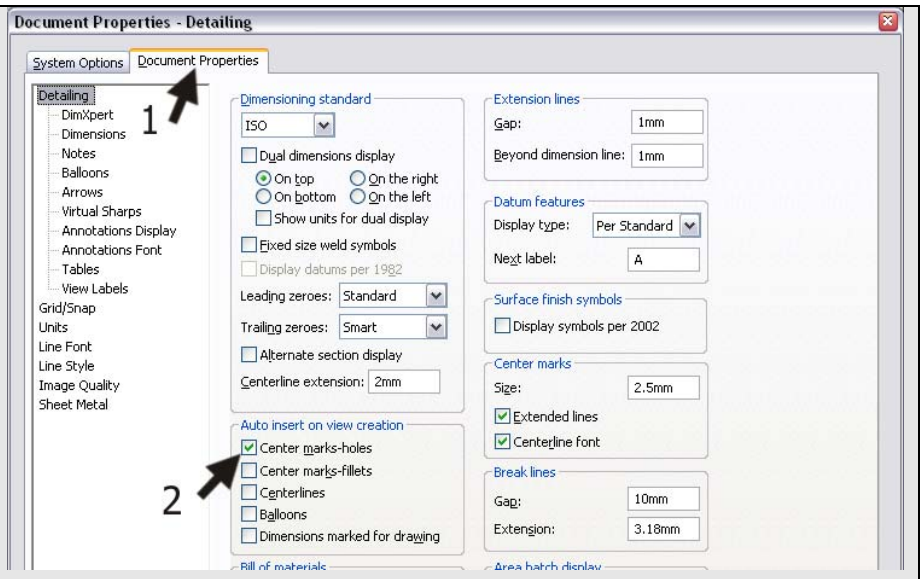
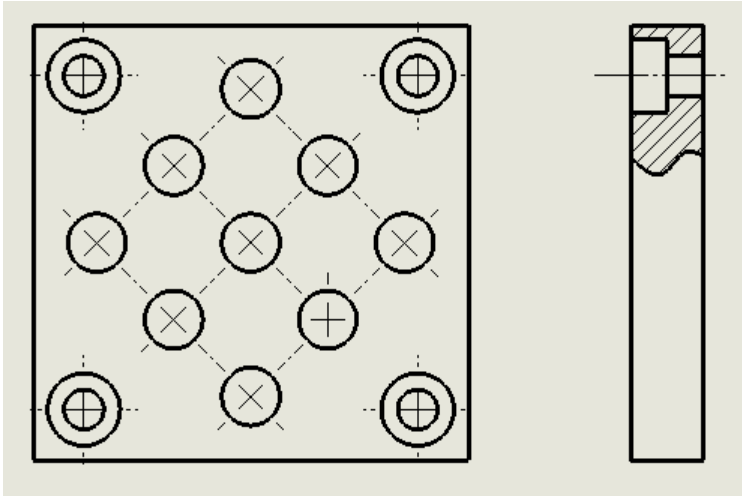
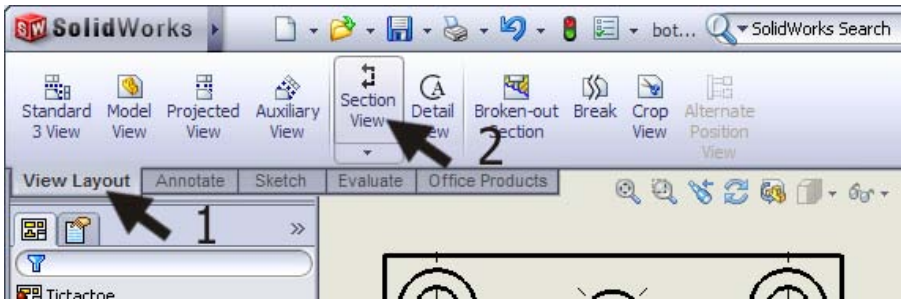
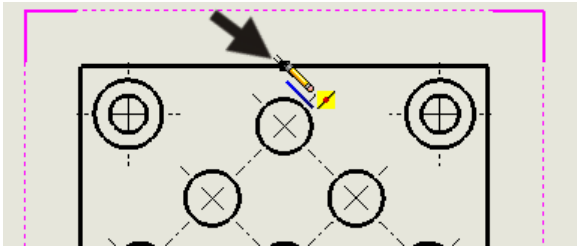
30	Next, we will place part numbers in the drawing. 1. Select the side view. 2. Click on 'AutoBalloon' in the CommandManager.	
31	1. Select the option 'Top' in the 'Balloon Layout' tab in the PropertyManager. 2. Select the option 'Balloon Faces'. 3. Click on OK.	

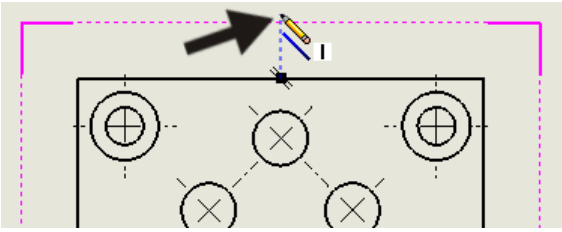
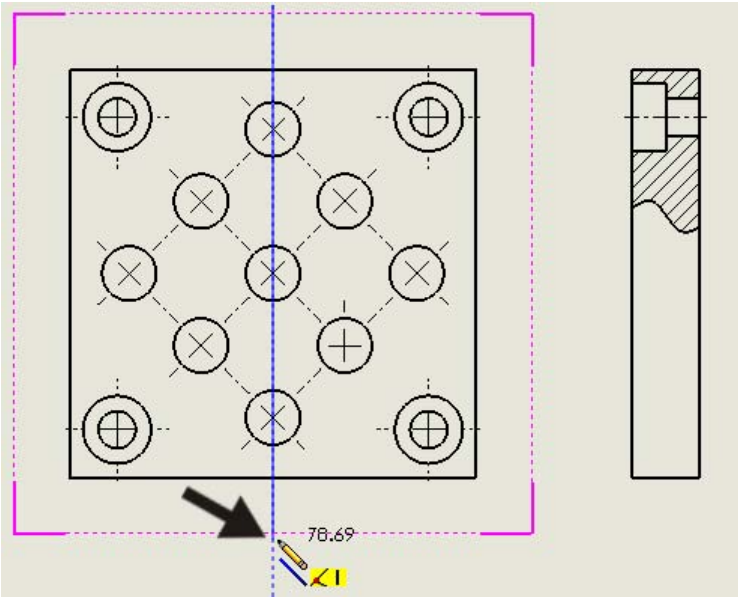
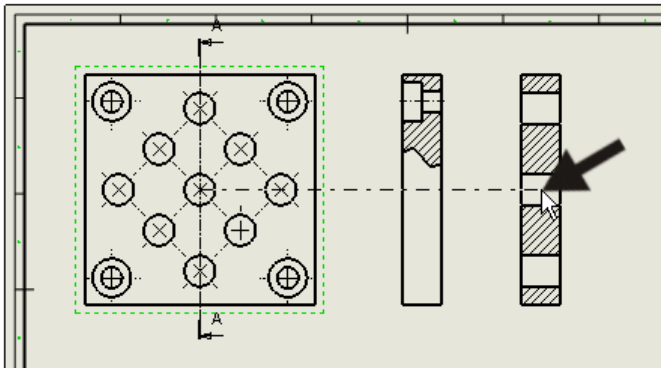
32	Now, you can place the parts numbers in their positions. Click on every parts number. You can drag the number balloon as well as use the arrow now. When you do not put the point of an arrow on a line of a figure, the arrowhead will automatically turn into a dot. Try to position the parts numbers as in the illustration on the right.	
33	The composition drawing is now ready, except for one thing: you have to fill in your name in the title block. 1. Right-click somewhere in the drawing (not on a view). 2. Select 'Edit Sheet Format' in the menu. The drawing now temporarily disappears, and you can change the items in the title block.	
34	1. Double-click on the text 'Name:', and fill in your own name. 2. Click on OK.	

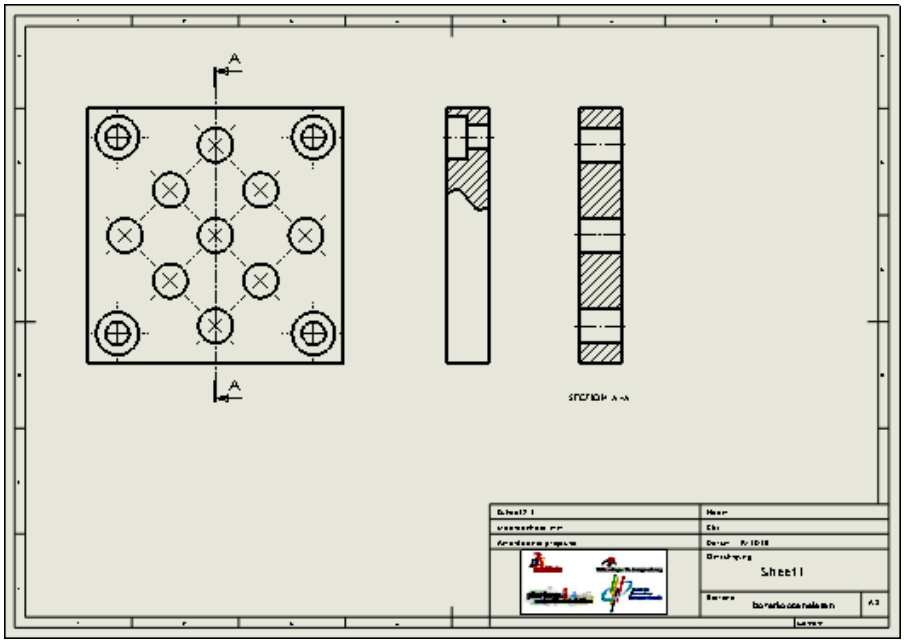
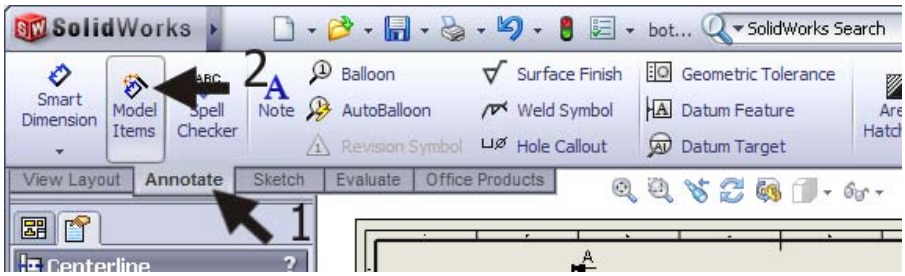
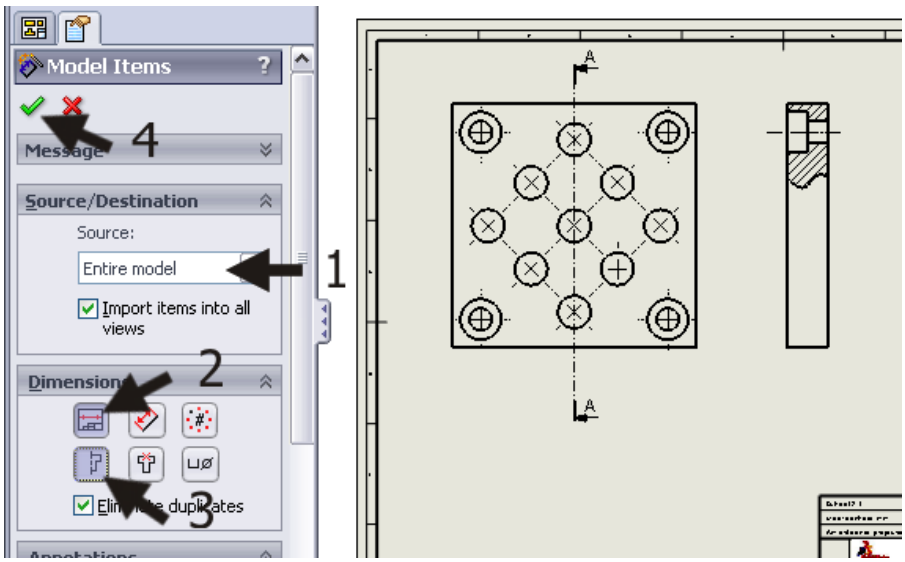
35	1. Right-click in the drawing again. 2. Select 'Edit Sheet' in the menu. The drawing reappears.	
36	Save the file as: Tictactoe.SLDDRW.	
37	Next, we will make a single drawing of the top plate. We will first add a new drawing. Click on Add sheet at the bottom of the screen.	
	Tip!	We use Add Sheet to add a drawing sheet within the same file. Of course, we could have created a second file, but in this way we will keep drawings together and provide a better overview.
38	When the menu of step 39 does not appear by itself, right-click somewhere in the drawing and select 'Properties'.	

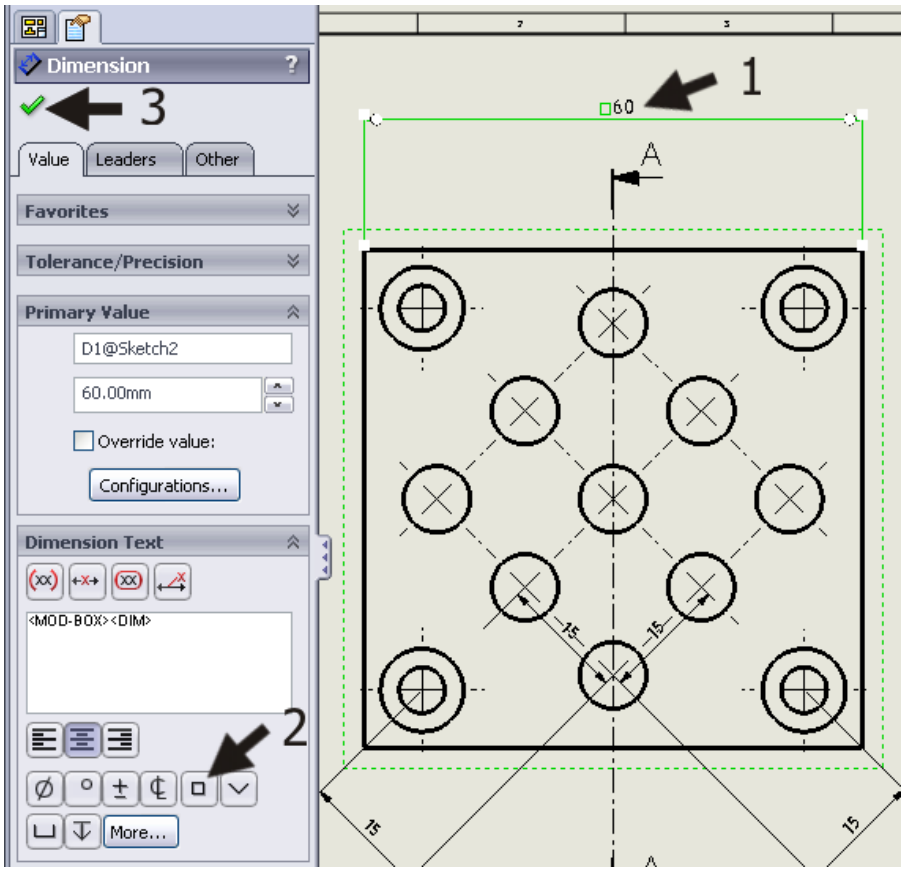
39	Most of the settings for this drawing will be the same as the settings for the first drawing. Therefore, there is not much we have to change. 1. Change the name of the sheet to 'Slab-top'. 2. Click on OK.	
40	We will use the Task Pane to place a view on the drawing board Click on the tab 'View Palette' in the Task Pane.	
41	The views you see in the 'View Palette' bar, are the ones that are in the assembly. To load the top plate, click on the Browse ('...') button at the top of the Task Pane.	

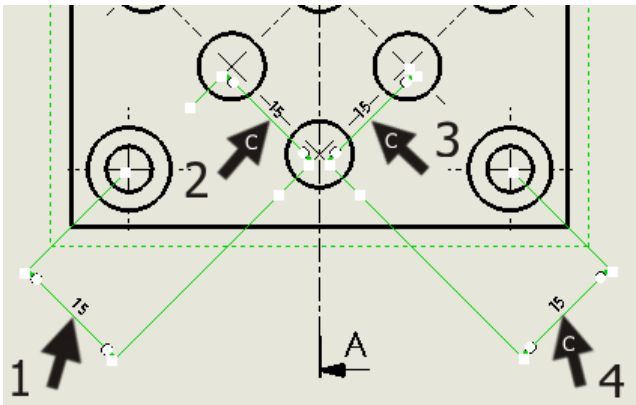
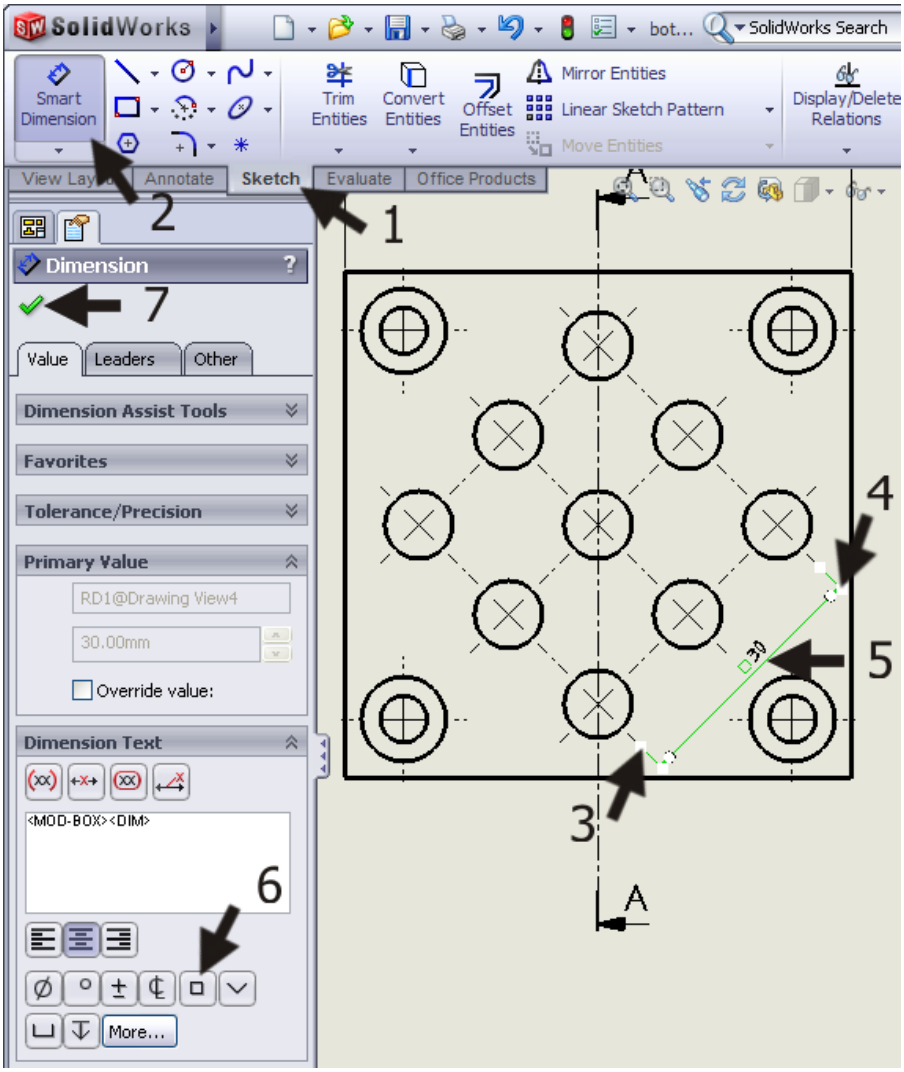
42	1. Click on the part 'Slab.SLDPRT'. 2. Select the configuration 'Top'. 3. Click on 'Open'.	
43	In the View Palette (on the right of the screen) the views of the top plate are visible now. 1. Drag the Top-view to the drawing sheet. 2. Click to the right of the top view to place a side view. 3. Click on OK in the PropertyManager.	
	Tip!	Notice that the Center Marks of all holes have been added to the view automatically. In the drawing of an assembly, SolidWorks does not do this automatically. SolidWorks does this, however, in a drawing of a part, if this feature is set. SolidWorks has dozens of settings for creating drawings. We always pick the standard settings, but it is possible that the settings on the computer you are working on have been changed. Some features may look of even work differently. If you want to have a look at all the possible settings, click on Options in the Standard Toolbar.  Click on the 'Document Properties' tab in the menu. Here, there are all types of settings, including the option to place Center Marks automatically.

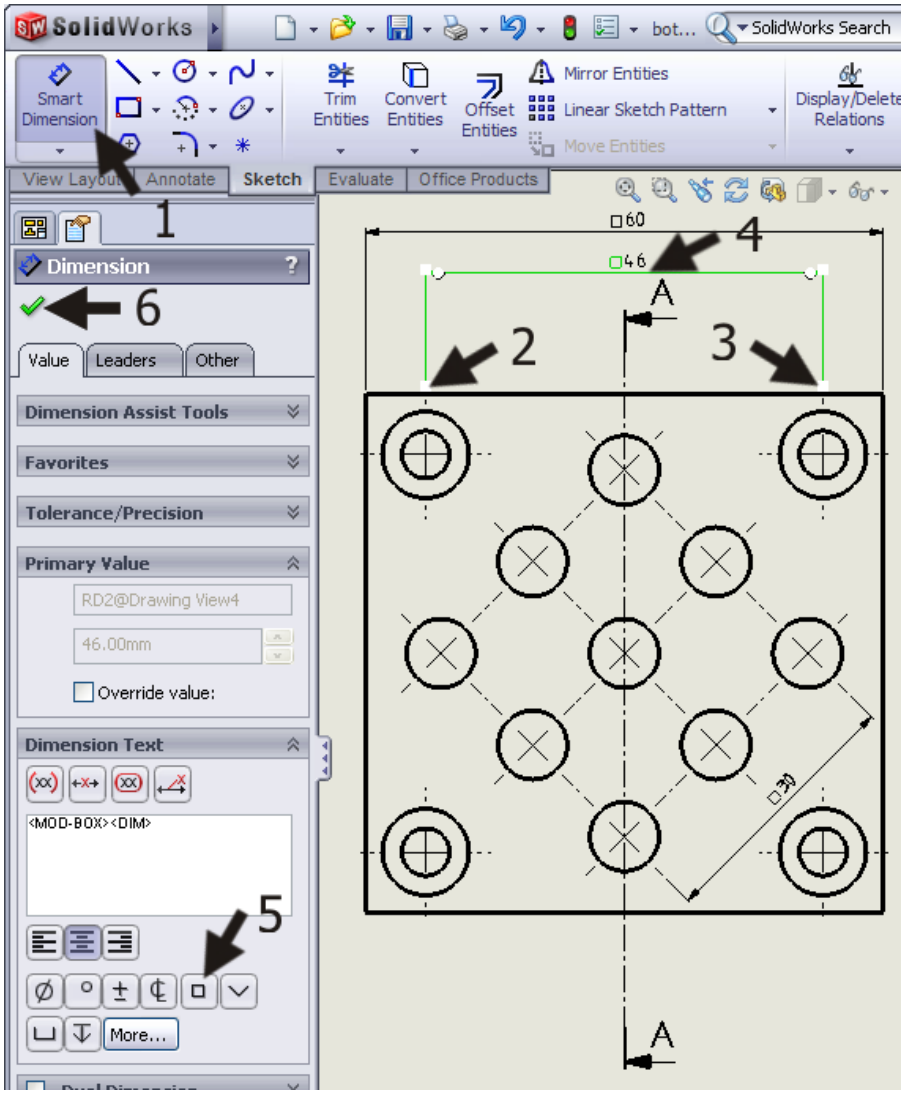
		
44	Break open the side view so you have a clear view of the counter bore hole. Can you remember how to do this? Check steps 11 to 15 of this tutorial. You did the same thing in the assembly! Put a centerline in the hole (look at step 23).	
45	We will draw a cross-cut now. 1. Click on 'View Layout' in the CommandManager. 2. Click on 'Section View'.	
46	Next, you have to draw the cross-cut line. Put the cursor directly above the middle of the top line in the top view but **do not click yet**!	

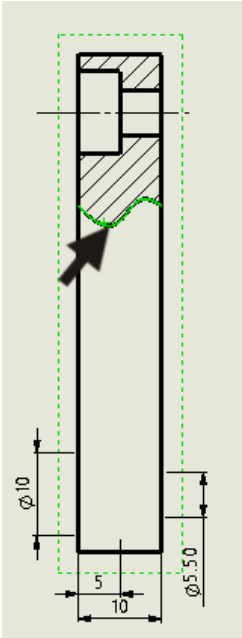
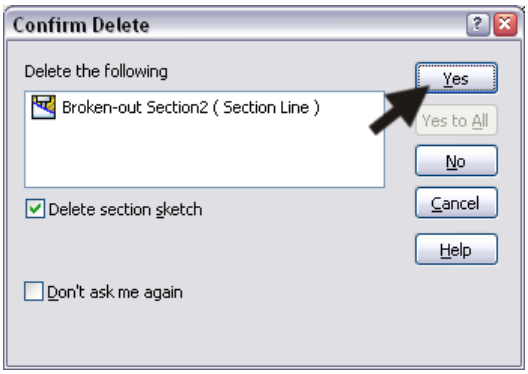
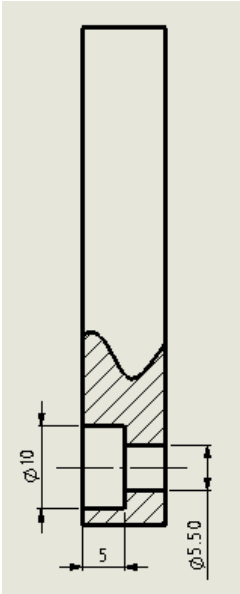
47	Move the mouse upwards. A blue dotted vertical auxiliary line appears. Click just above the view while this auxiliary line is still visible.	
48	Move your mouse straight down and click just below the view.	
	Tip!	Why could you not just click on the middle of the top line in the view at step 48? When you would have done this, the cross-cut line would have stopped at that point. The arrow and the letter to indicate the cross-cut section would appear in the middle of the drawing and that is just not what we want to have! It is not possible to change this feature later. We have created the line as described above, and it is possible to change the length.
49	Next click besides the side view to place the cross-cut drawing.	

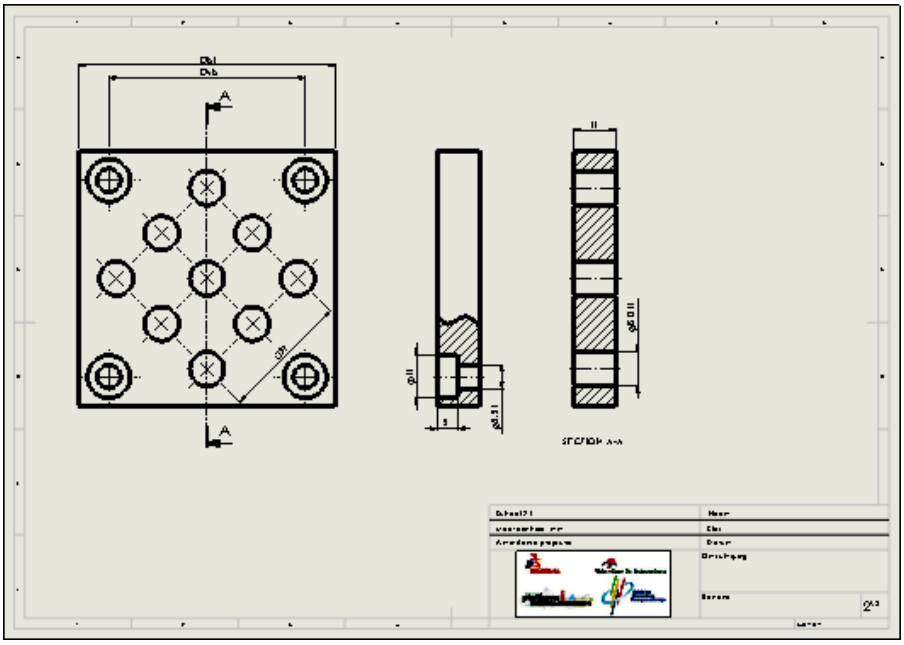
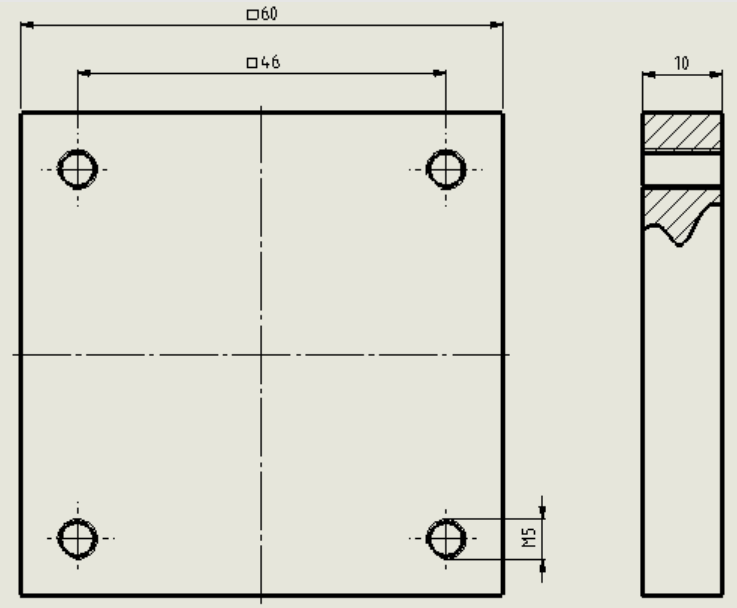
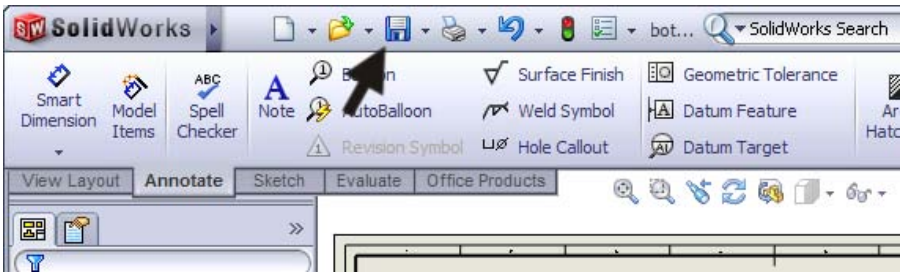
50	Move the views in such a way that they are placed on the sheet neatly. Add the centerlines in the cross-cut drawing.	
51	Finally, we have to add the dimensions to this drawing. 1. Click on 'Annotate' in the CommandManager. 2. Click on 'Model Items'.	
52	Set the following features in the PropertyManager: 1. Select 'Entire Model' in the 'Source' field. 2. Check the options Marked for Drawing in the 'Dimensions' tab. 3. Check the option Hole Wizard Profile. 4. Click on OK. The dimensions will now be placed in the drawing.	
	Tip!	With the Model Items command you will put parts of the model in the drawing. In this case we did that with the dimensions. We have checked two options: 1. **Marked for Drawing**: these are often all of the dimensions that you used when modeling the parts in sketches and when making the features.

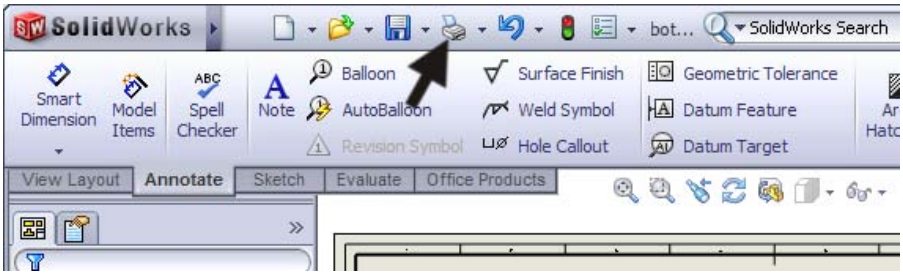
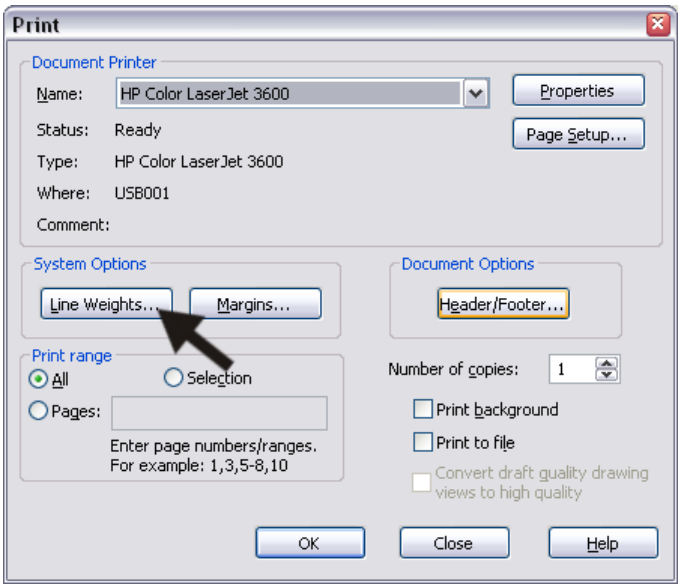
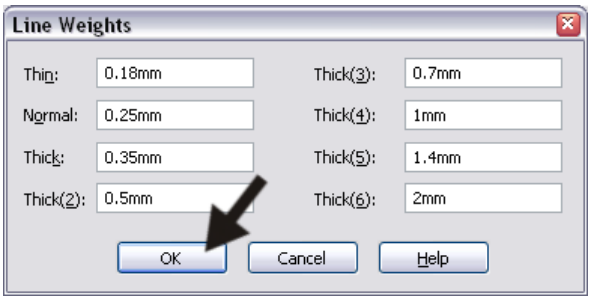
		2. **Hole Wizard Profile**: the shape of the hole you have made with the Hole Wizard. When adding dimensions to a drawing in SolidWorks, it is always very smart to start with Model Items. Although by doing so, the drawing is not finished yet! We will see that some dimensions are missing and other dimensions are in the wrong positions. You can change some items, but some of them must be deleted and replaced.
53	First, we will adapt the dimensions located at the outside edges of this part. 1. Select the dimension 60mm, and drag it (when necessary) a bit upwards, so it no longer crosses the centerline. 2. Click on the square in the 'Dimension Text' tab. The text in the field now changes to '<MOD-BOX><DIM>', and a square appears in the drawing in front of the dimension of 60mm. 3. Click on OK.	

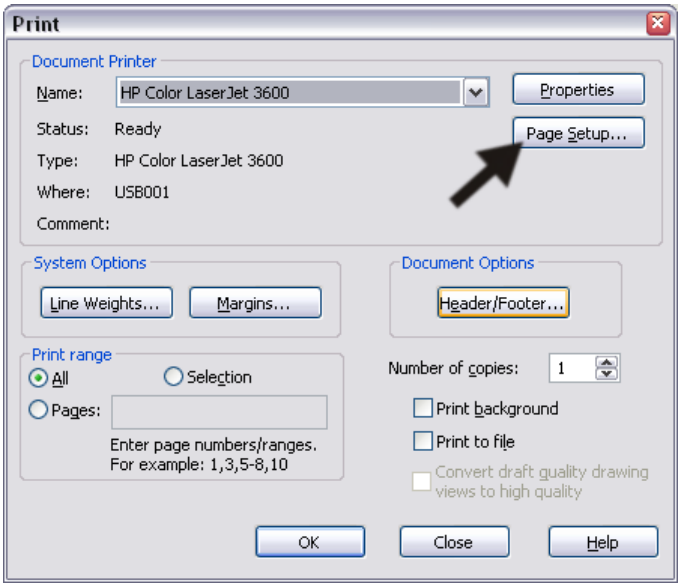
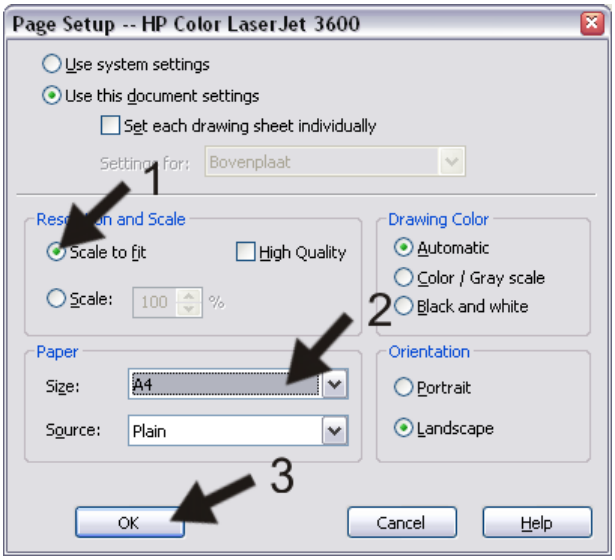
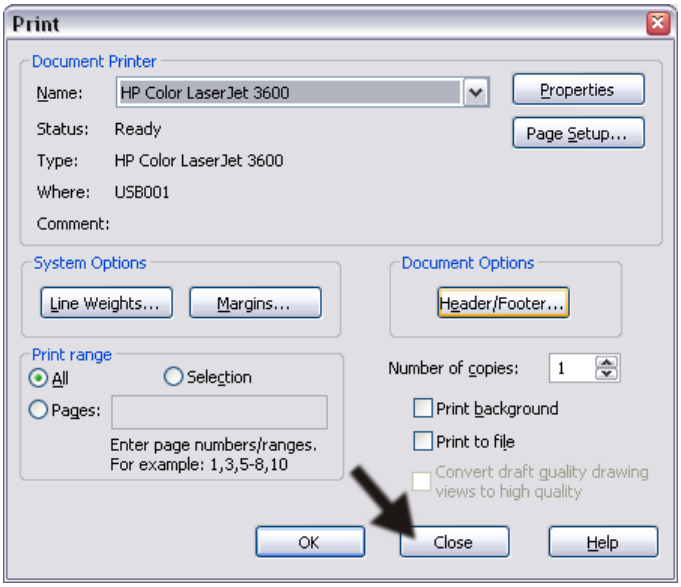
54	In the drawing, you will see the dimension of 15mm four times. We want to replace it with only one dimension of 30 mm. Select the four dimensions (hold the <Ctrl> key on the keyboard) and push <Del> (delete). You can also remove them one at a time.	
55	Next, we set the dimension of 30 mm. 1. Click on 'Sketch' in the CommandManager. 2. Click on Smart Dimension. 3,4 Click on the end of two centerlines. 5. Set the dimension. 6. The dimension is still selected (green). Click on the square symbol in the 'Dimension Text' tab in the PropertyManager. 7. Click on OK.	

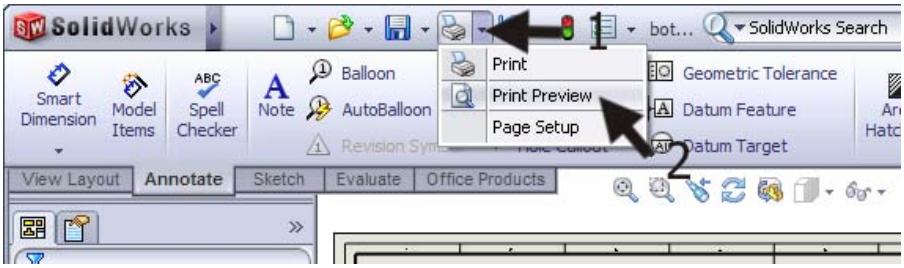
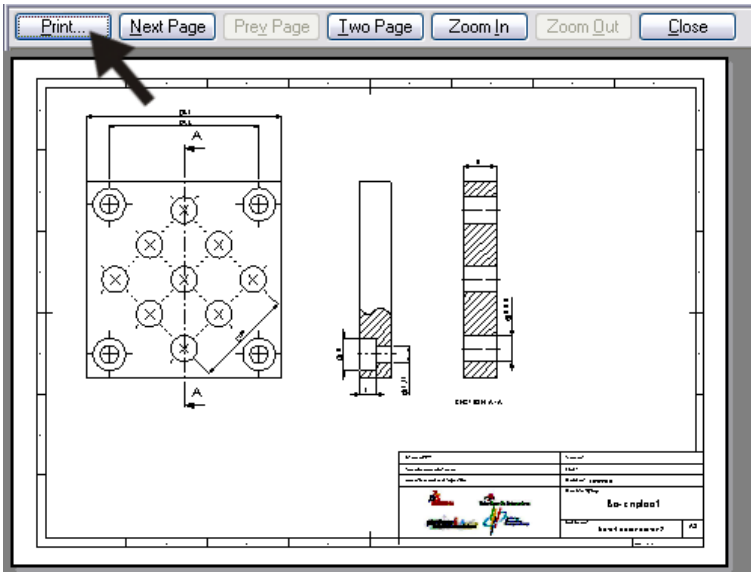
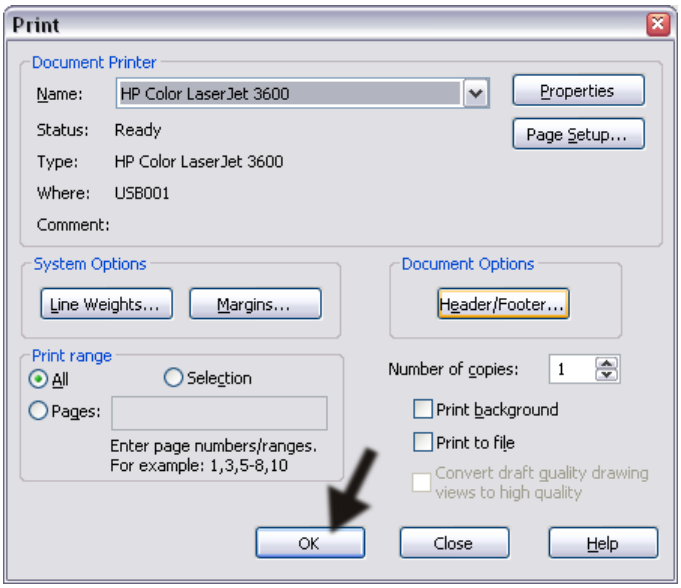
56	Next, we will put a dimension for the distance between the countersink holes: 1. Check if the command Smart Dimension is still active; if not, click on it in the CommandManager. 2,3 Click on the centerlines of the two upper holes. 4. Set the dimension. 5. Click on the square symbol in the PropertyManager. 6. Click on OK.	
	Tip!	You have seen that you can add dimensions very easily with Smart Dimension. Please realize that there is a difference between the dimensions that you import from a model and the dimensions that you add yourself: Imported dimensions are 'real' dimensions (driving dimensions). When you double-click and change them, the model will change as well! Added dimensions are extracted dimensions (driven dimensions). You can change the value of the text in the PropertyManager, but it will not have any influence on your model.
	Work plan	We will change the dimensions of the side view now. You can see that the dimensions of the countersink hole are set below the drawing and not at the point where we made the cross-cut. It may be different in your drawing: this depends on the order in which you have made the holes while modeling: When the dimensions are in the same position as they are in the drawing, you can do two things: 1. Delete the lower dimensions and add the one at the top. 2. Delete the cross-cut section and renew this at the bottom.

		Because we want to work with imported dimensions as much as possible, we will choose the second option.
57	First we remove the cross-cut from the view. 1. Click on the cross-cut line in the view. 2. Push the <Del> (delete) key on the keyboard.	
58	Click on 'Yes' in the menu that appears.	
59	Next, draw a Broken-out Section at the lower side of the view. Can you remember how this is done? If not, check steps 11 to 15. Put a centerline in the hole. Remove the lower dimension 10 (in the drawing at step 57 you can still see this dimension).	

60	Fill in your name in the title block. Can you remember how this is done? If not, check steps 35 to 37. The drawing of the top plate is now done.	
	Work plan	Next, we have to make the drawing of the bottom plate. This actually is a simplified version of the top plate. A separate cross-cut of this drawing is not necessary. Look at the drawing below.  Draw this one yourself! You can follow steps 37 to 60 if you need them.
61	Now, you have made three drawings. Save the file.	

65	1. Click on Print in the Toolbar.	
66	Ask your teacher for the exact settings for the 'Print' menu. We only show you a few important settings. Click on 'Line Weights...'.	
67	Check to make sure that the line thickness is set like in the illustration on the right. Click on OK.	

68	Click on 'Page Setup...' in the 'Print' menu.	
69	Again, ask your teacher for the correct setting for this menu. 1. Check the option 'Scale to fit'. The drawing will print at its maximum size for the size of paper used. 2. Select the format of the paper. 3. Click on OK.	
70	Click on 'Close' in the 'Print' menu.	

71	Click on 'Print Preview' in the Standard Toolbar.	
72	You will see a view how the drawing will be printed. Check to make sure everything is OK and click on 'Print...'.	
73	You will return to the 'Print' menu. Click on OK.	
	What are the main features you have learned in this tutorial?	In this tutorial you have created your first drawings with SolidWorks. You have learned how to extract drawings from a model. What else did you do? - You changed the settings of the drawing sheet. - You placed views according to the American or European projection method. - You made cross-cuts.

		- You added threads in a drawing. - You added part numbers and a parts list in the assembly. - You imported and positioned dimensions. - You filled in the title block. You have used the most important features of the drawing commands now, so you will be able to create most drawings. In Tutorial 10 we will make some more drawings.

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 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

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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