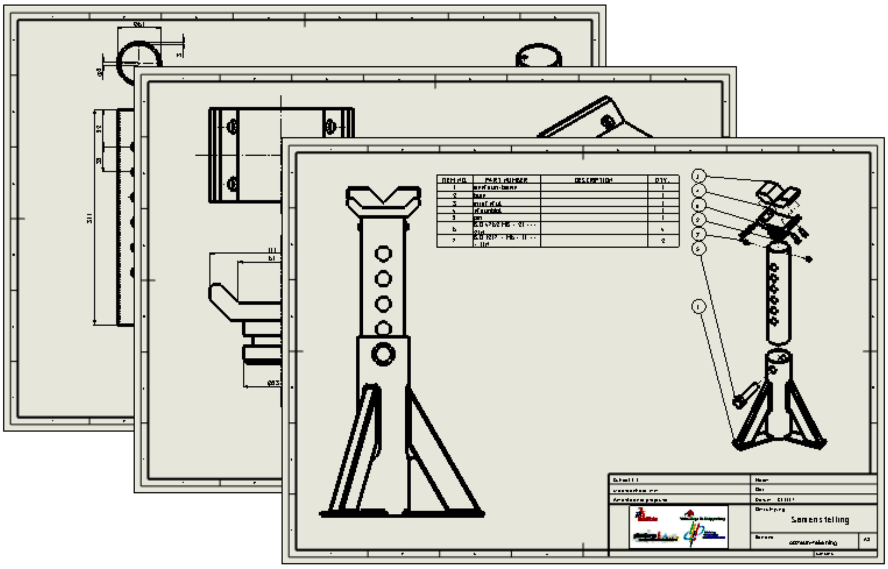
SolidWorks® Tutorial 10

DRAWING OF THE AXLE SUPPORT



Preparatory Vocational Training
and Advanced Vocational Training

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

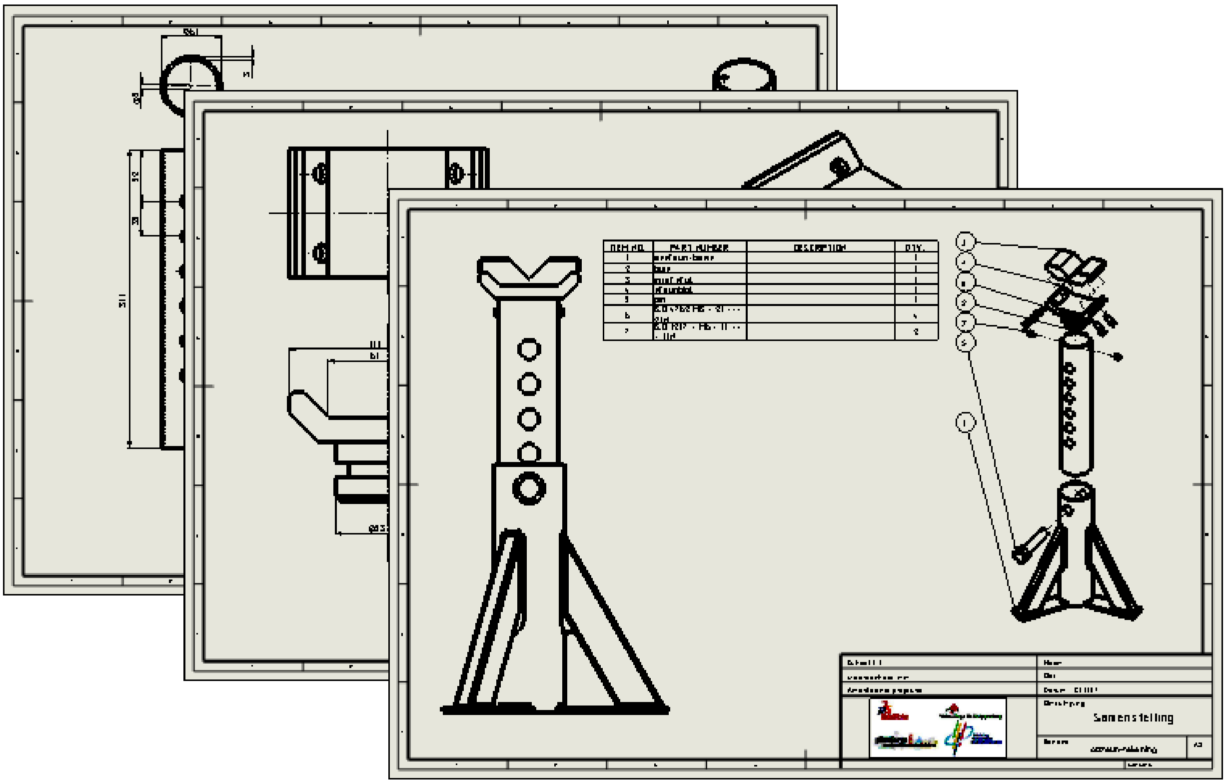
SolidWorks Benelux developed this tutorial for self-training with the SolidWorks 3D CAD program. **Any other use of this tutorial or parts of it is prohibited.** For questions, please contact SolidWorks Benelux. Contact information is printed on the last page of this tutorial.

Initiative: Kees Kloosterboer (SolidWorks Benelux)
Educational Advisor: Jack van den Broek (Vakcollege Dr. Knippenberg)
Realization: Arnoud Breedveld (PAZ Computerworks)

Axle Support, the Technical Drawing

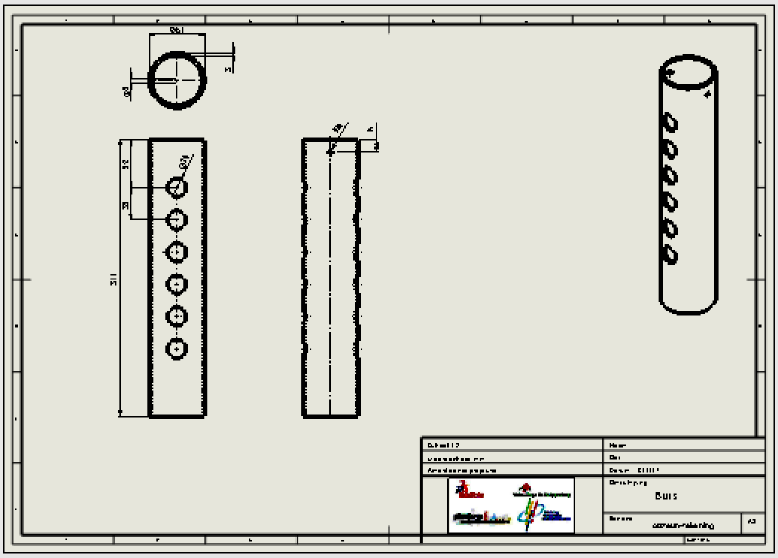
In this tutorial we will make some technical drawings of the 3D model of the axle support that we created in Tutorial 9. This tutorial is designed to continue with the files you made in Tutorial 9. If you did not finish the previous tutorial or lost the files, ask your teacher about them.

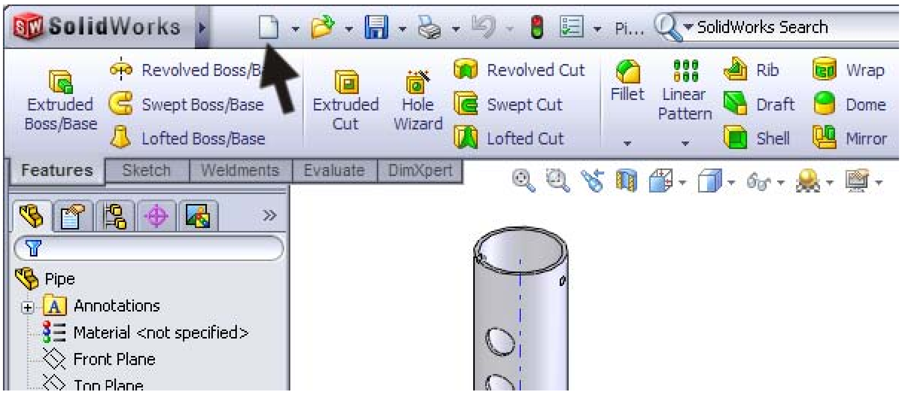
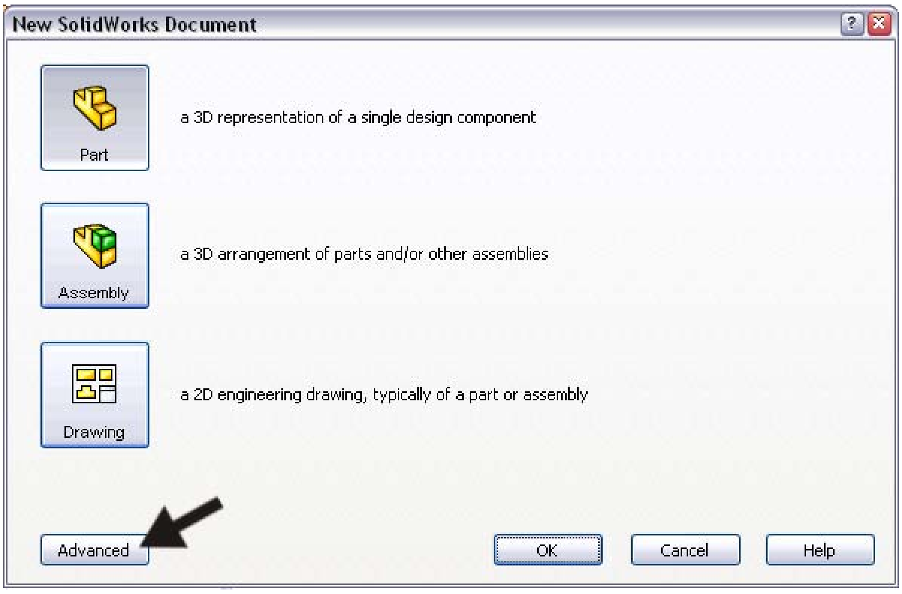
Creating a 2D drawing is not very difficult. We will show you a number of examples of single part and assembly drawings. Also, we will show you how to make an exploded view.

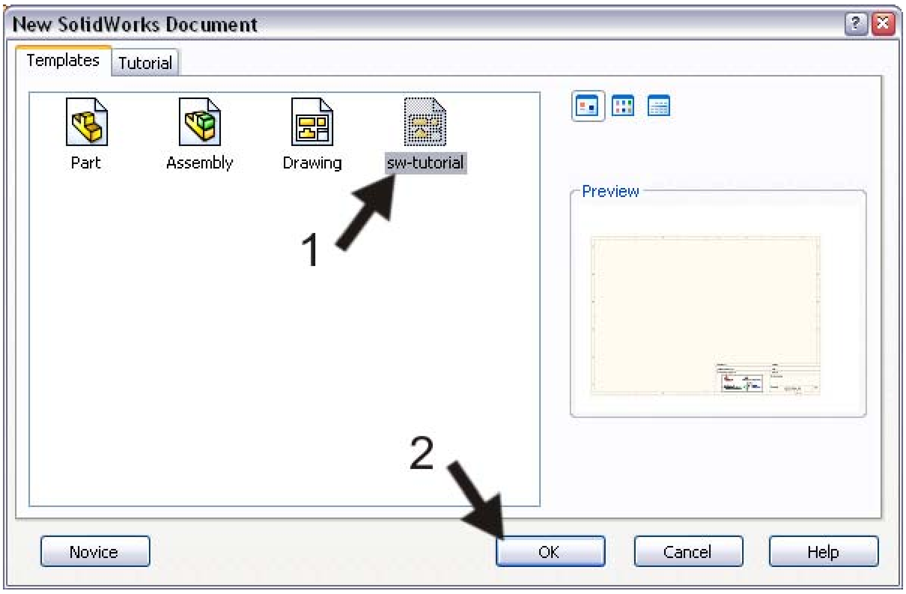
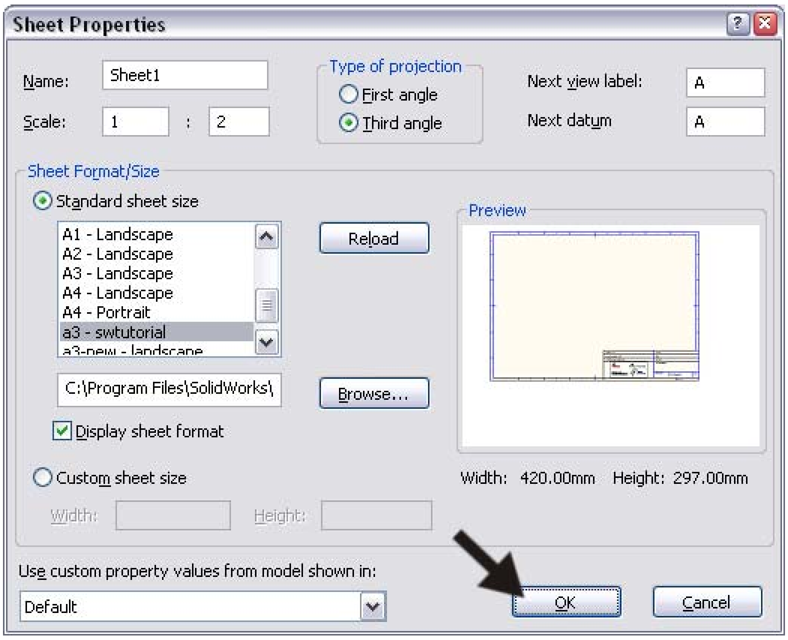
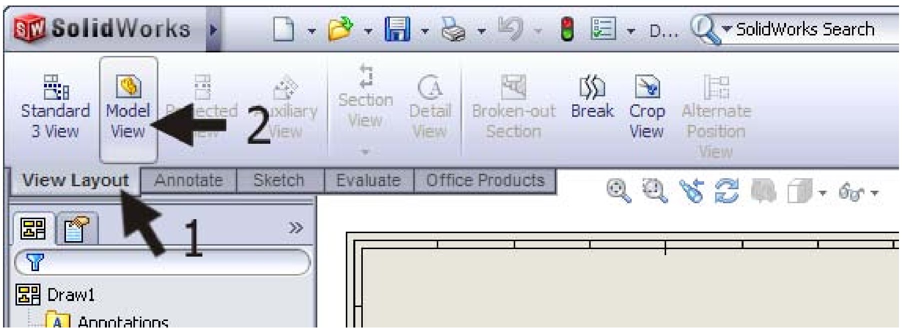


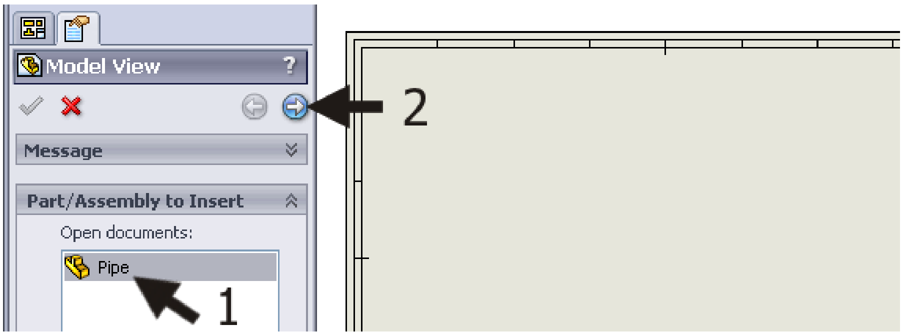
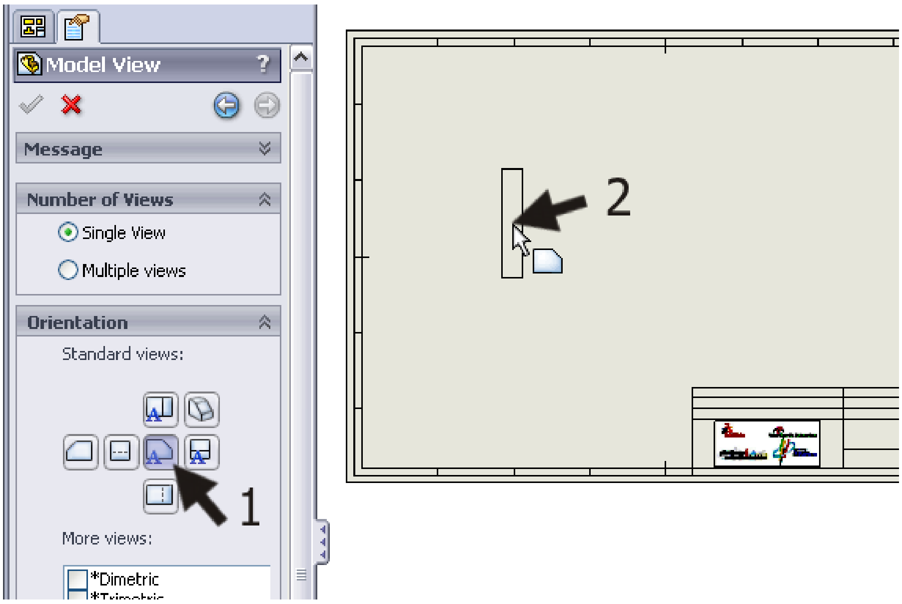
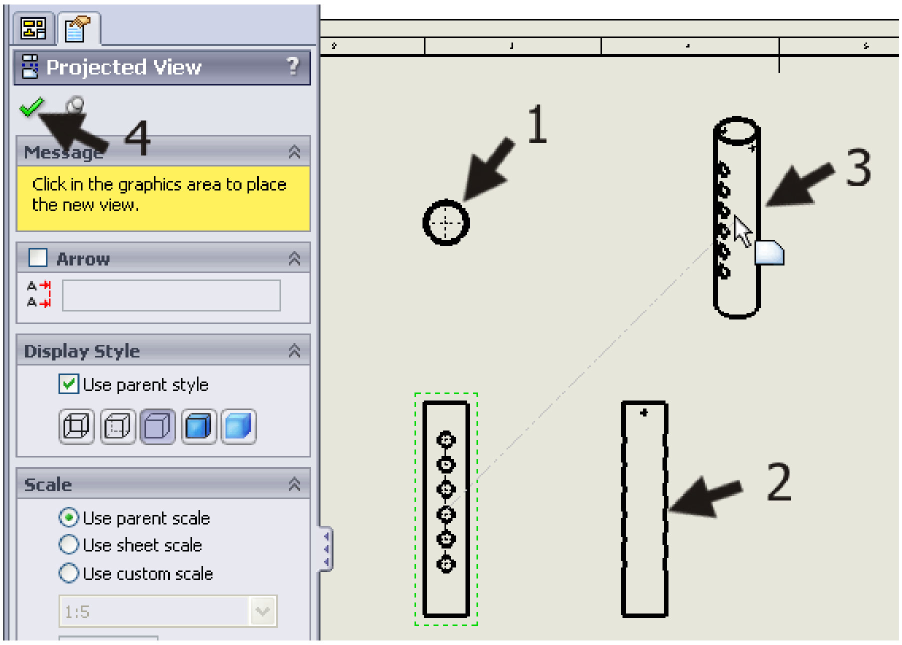
Work plan

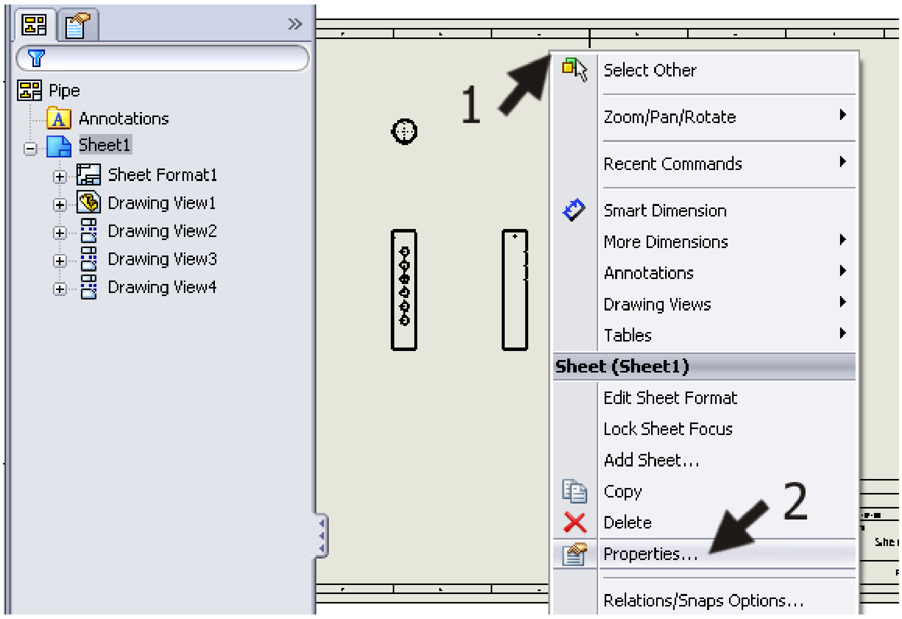
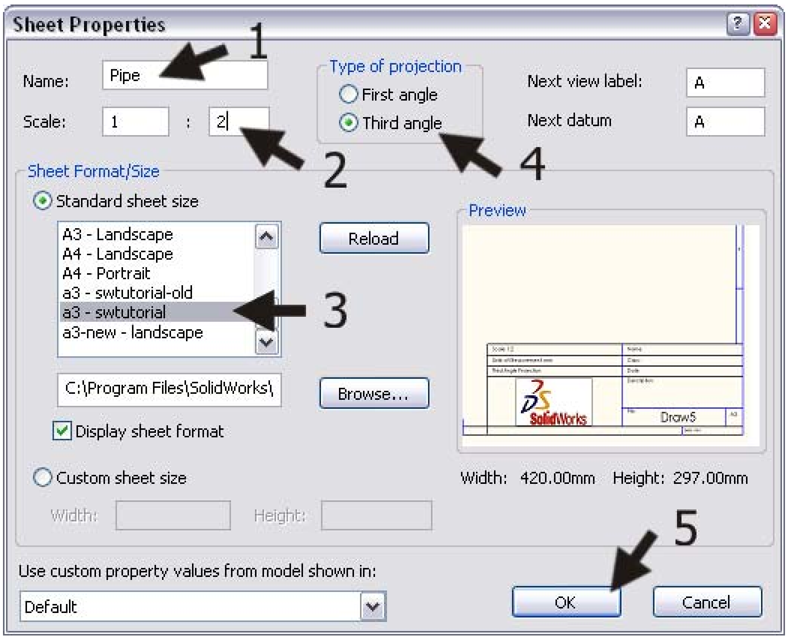
The first part is the inner tube of the axle support. We will put three views on the drawing sheet and an isometric view.

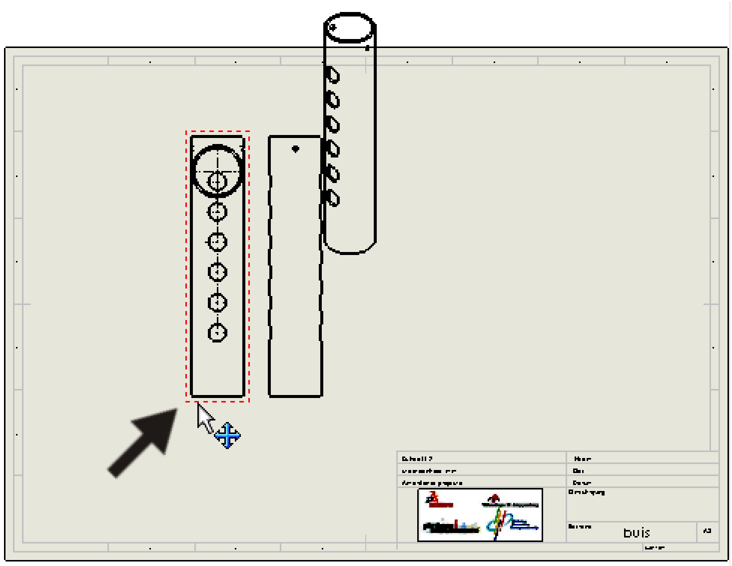
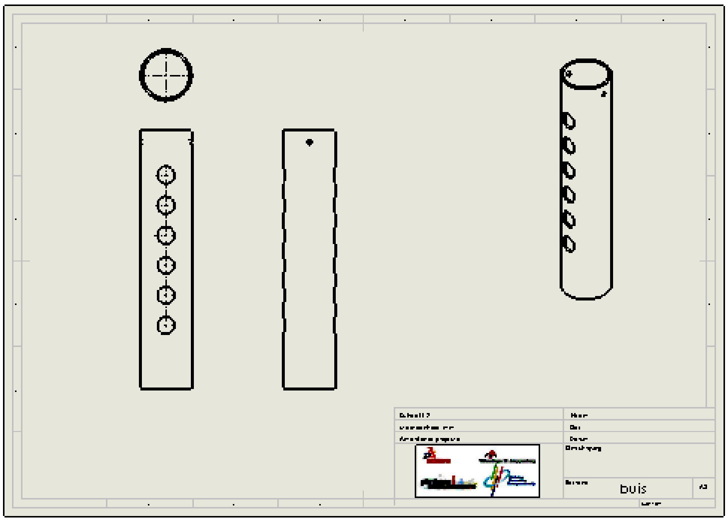
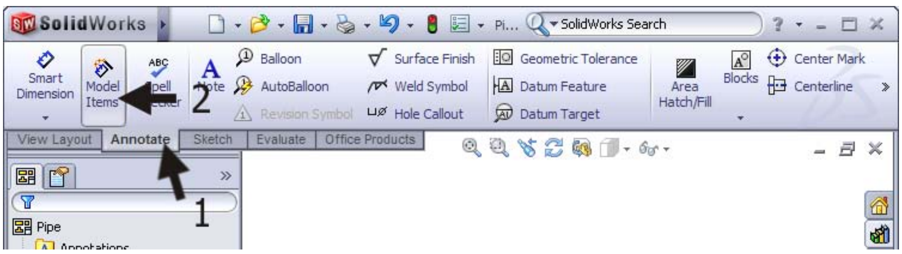


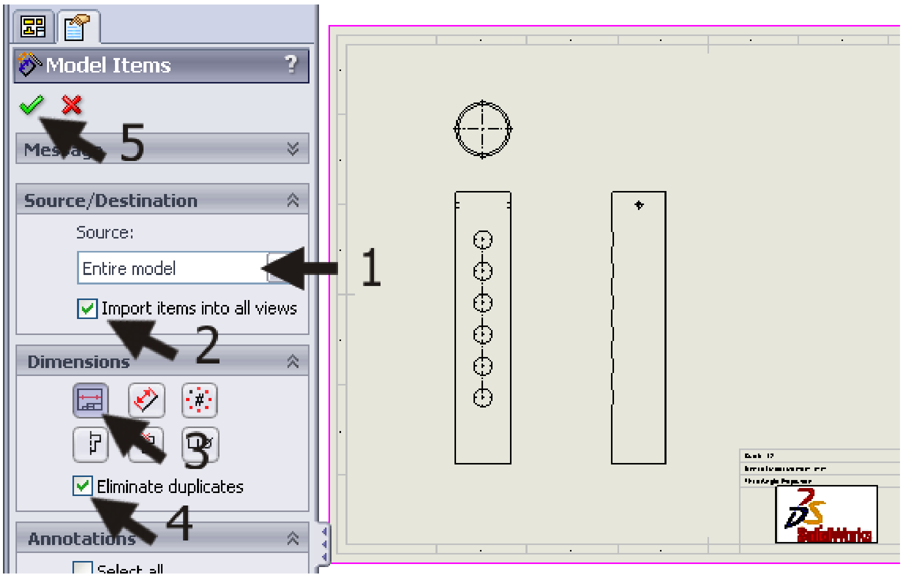
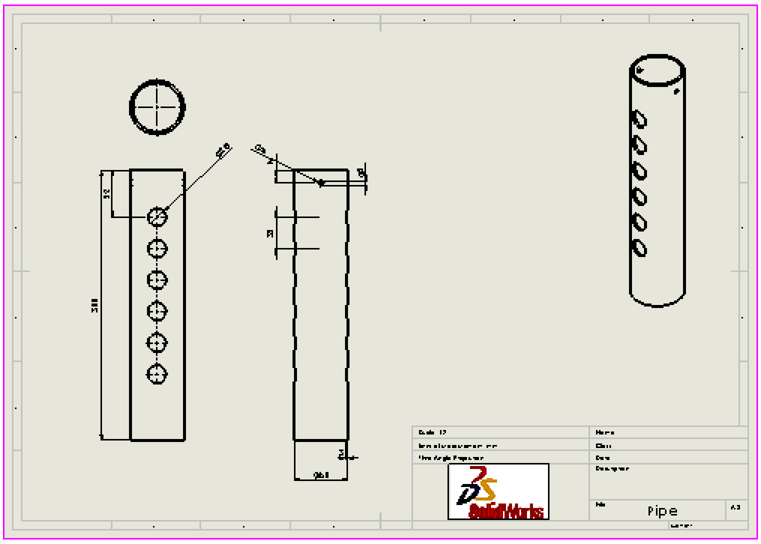
1	Start SolidWorks and open the part pipe.SLDPRT. You created this file during the last tutorial.	
2	Click on New in the Standard Toolbar to open a new file.	
3	Click on 'Advanced' in the pop-up menu.	

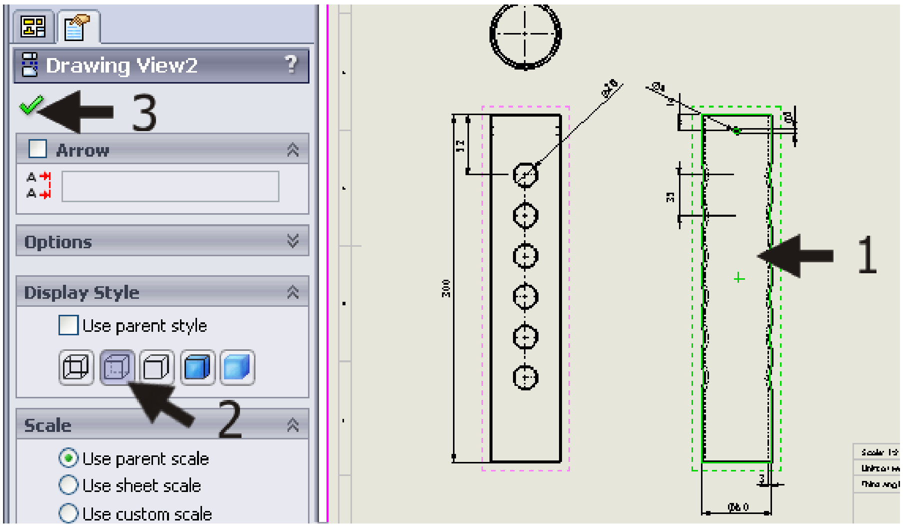
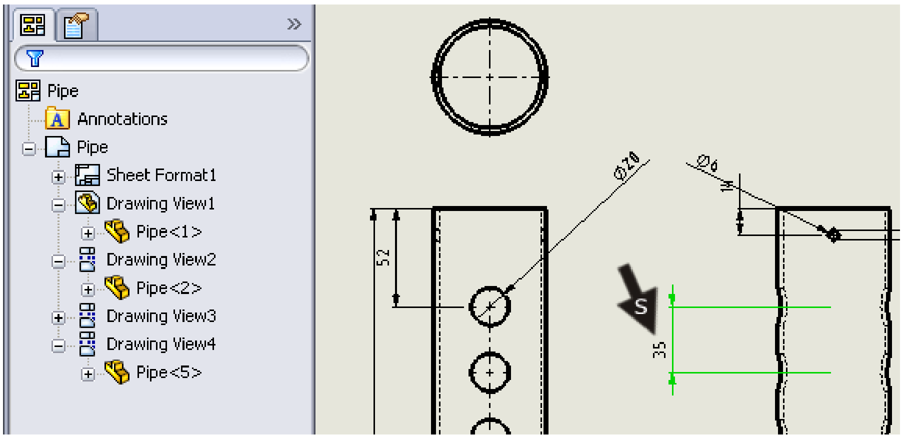
4	Select the file 'sw-tutorial' to be your template and click on OK. If this file does not exist, ask your teacher for it. In this file we have made a number of standard settings, so you can start building a proper technical drawing. If you are working at home, you can simply download the file sw-tutorial.DRWDOT and put it in the folder: C:\Program Files\SolidWorks\data\templates.	
5	The menu shown on the right might appear. If the menu appears, click on OK. We will get back to this later.	
6	An empty drawing sheet will appear. If the command 'Model View' does not start automatically, click on 'Model View' in the CommandManager.	

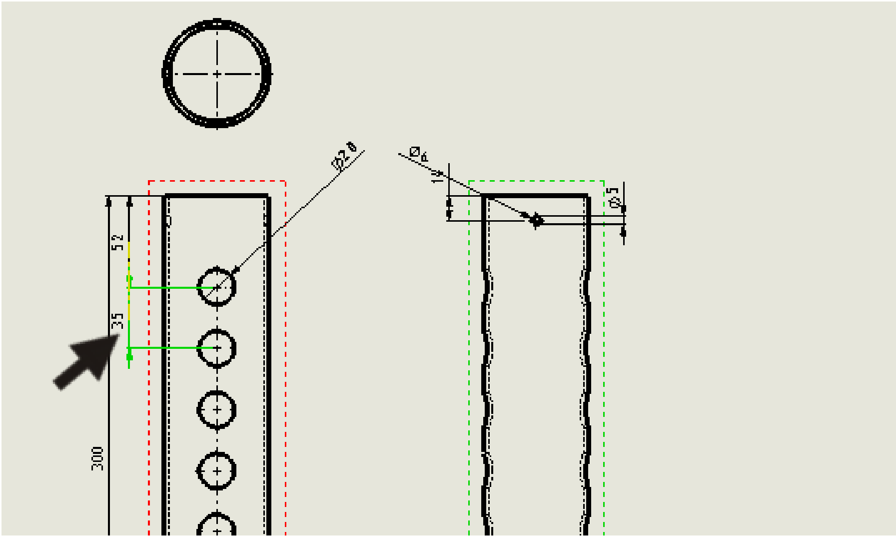
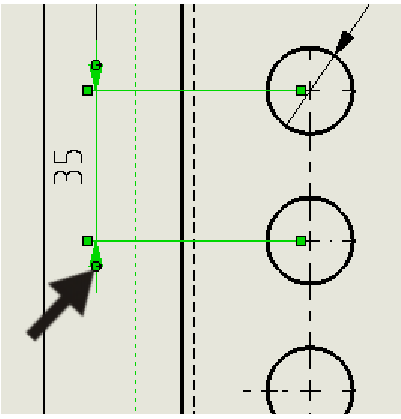
7	Set the first view now: 1. Click on the part 'Pipe'. 2. Click on Next.	
8	Make sure the settings in the PropertyManager match the ones on the right. 1. Select Front for the first view. 2. Put it somewhere on the sheet.	
9	After you have positioned the first view, the 'Projected View' command will start automatically. Move your cursor around the front view that you put in first. Click three times to set the three views as shown. Click on OK. If the 'Projected View' command does not auto-start, click on 'Drawings' in the CommandManager and after that on 'Projected View'.	
	Tip!	There is another method for placing views in a drawing. You can use the Task Pane command. You have done this before in Tutorial 6 (Step 41). As always in SolidWorks: use the method that you prefer!

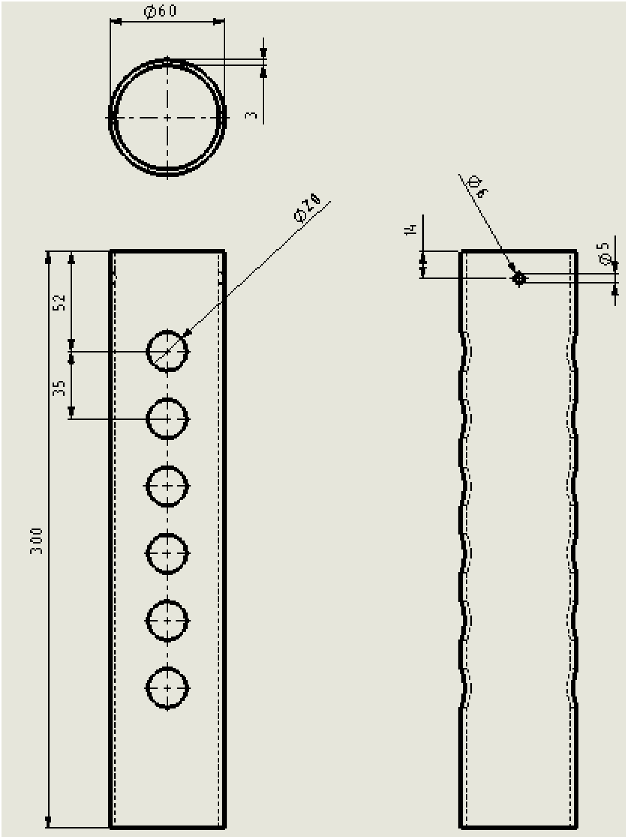
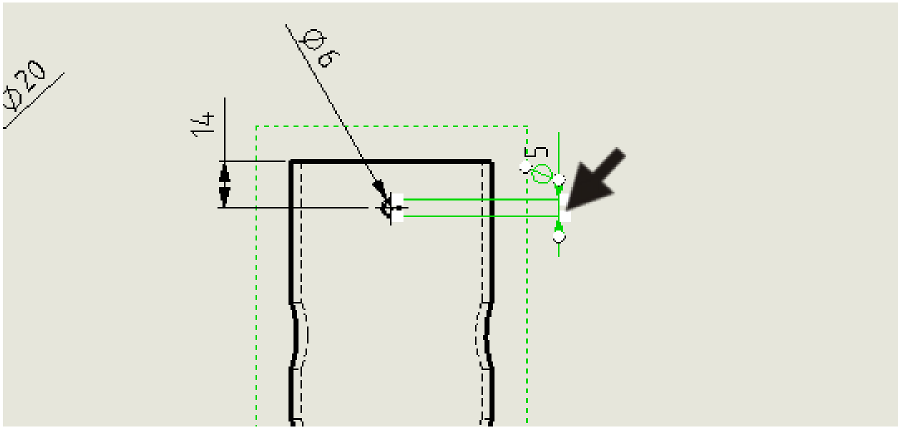
10	To change the main settings of the drawing, right-click at a random point on the drawing sheet (not on a view). Then, select 'Properties...'.	
11	Make sure to check the following settings: 1. 'Name' the drawing: 'Pipe'. 2. Select a 'Scale' of '1:2'. 3. Paper size is 'a3-swtutorial'. When this file is not available, ask your teacher for it. 4. Select 'Third Angle' (or American Projection, mostly used in the Netherlands) or 'First Angle' (European Projection, mostly used in Belgium) at 'Type of projection'. 5. Click on OK.	

12	The views intersect now. To change their positions, drag (click and hold your mouse button and move your mouse) the dotted frame that is visible around the view by moving your cursor over it.	
13	Position the views as shown on the right.	
14	Click on 'Annotate' in the CommandManager, and then on 'Model Items'.	

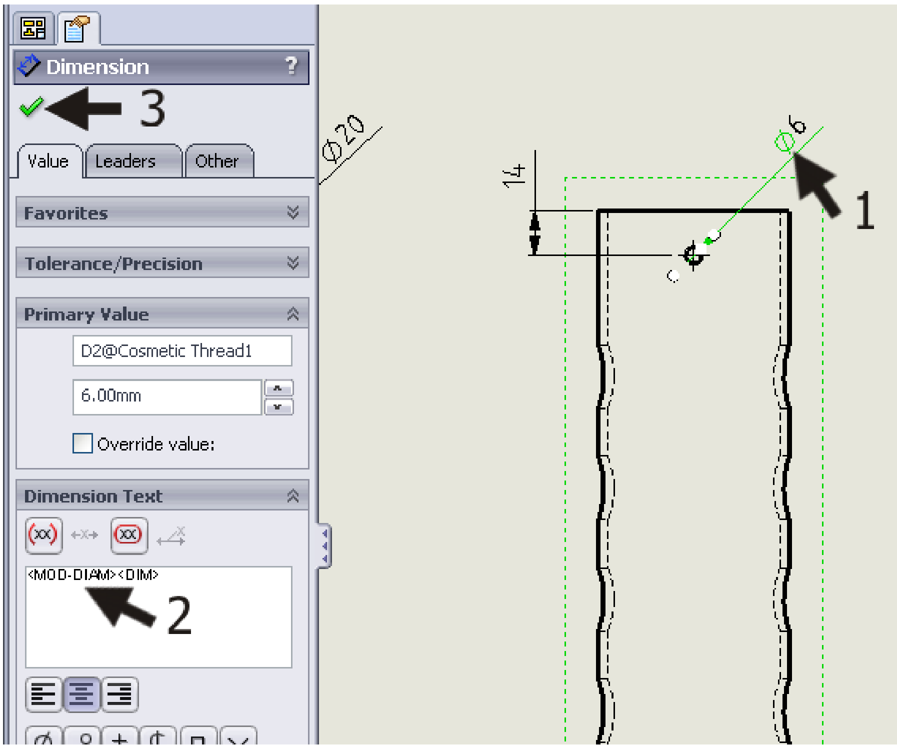
15	Set the following features in the PropertyManager: 1. Select the option ‘Entire model’ at ‘Source/Destination’. 2. Check the option ‘Import items into all views’. 3. Select the first option: Marked for drawing under ‘Dimensions’. 4. Check the option ‘Eliminate duplicates’. 5. Click on OK.	
16	The dimensions will now be displayed in the drawing.	
	Tip!	While modeling a part you will set a number of dimensions. You do this in sketches and in features. What we did just now, is nothing more or less than copying these dimensions onto the drawing. So SolidWorks did not come up with something by itself. When you do a sloppy job while modeling, it will show up in the dimensions on the drawing. Luckily, you can remove or change the dimensions manually and you can also add them to the drawing. In the following few steps, we will show you how.

17	We will show the invisible lines (dotted lines) in the drawing now. 1. Click on the side view. 2. Select the second option Hidden lines visible under 'Display Style' in the PropertyManager. 3. Click on OK.	
18	Do the same for the front view.	
19	Next, we want to put a number of dimensions in one of the other views. For example: the dimension between the holes in the tube (35mm) is now in the right-side view but we would rather show it in the front view. 1. Drag the size from the right-side view holding the <Shift> key. 2. Release the size somewhere in the front view. 3. Then, release the <Shift> key on your keyboard.	

20	Next, drag the size to the right position. Make sure the size is aligned with the size '52' which is above it. While dragging you can see (yellow) auxiliary lines that indicate if the sizes are actually aligned.	
21	To shift the arrows on both sides of the auxiliary lines, do the following: 1. First, select the size by clicking on it. 2. Click on the round dot you see besides the arrow. The arrows will now move to the inside of the line.	

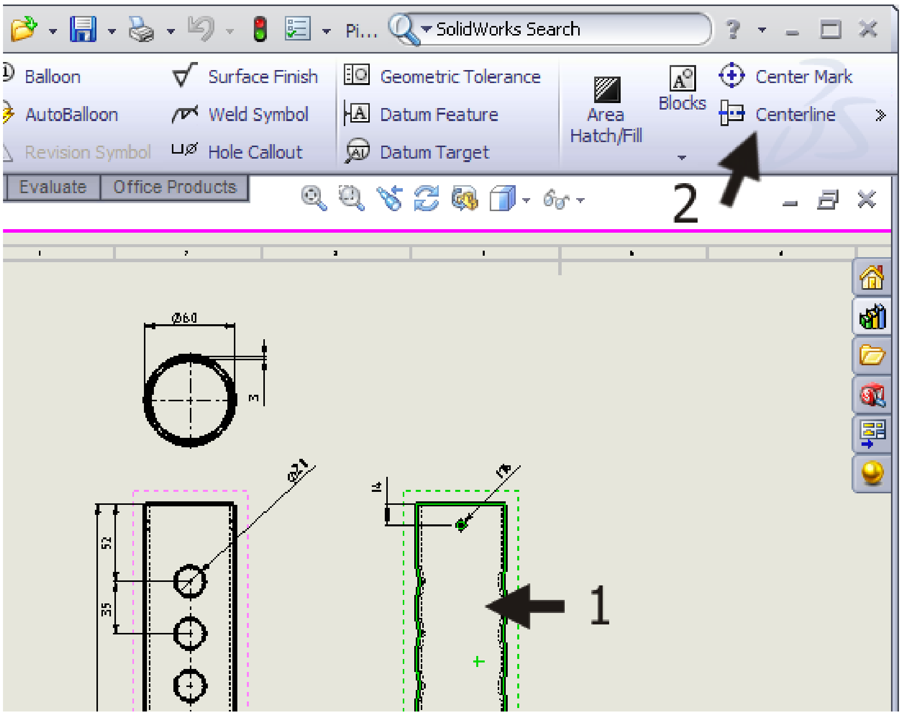
22	Move a few more dimensions like you did in Step 19: The tube diameter (Ø6) and the thickness of the material (3) are moved to the top view. The drawing must look like the illustration on the right.	
23	The size Ø5 can be removed. 1. Click on the size. 2. Push the <Del> delete key on the keyboard.	

<table>
<tr>
<td>24</td>
<td>
We want to change the size Ø6 to M6, because it is a screw thread.

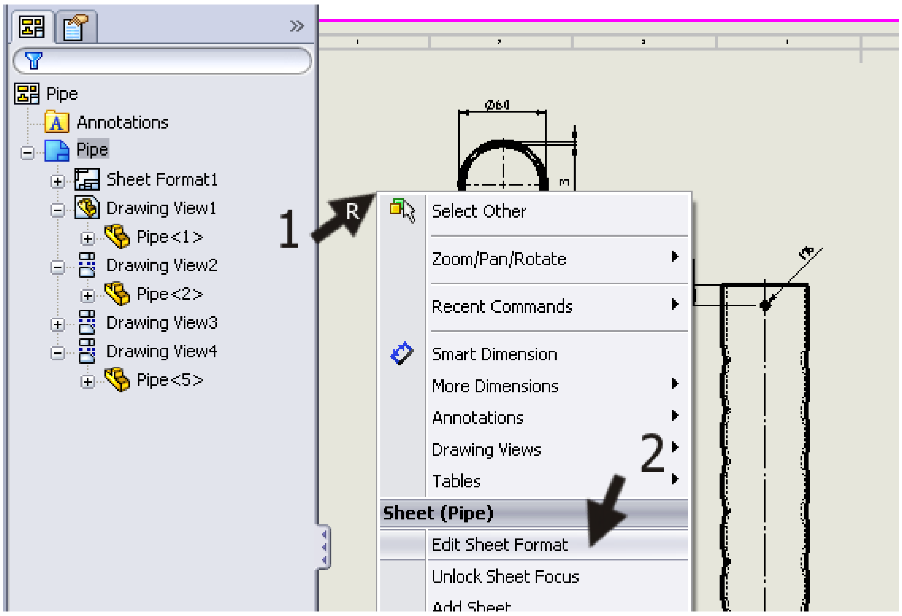
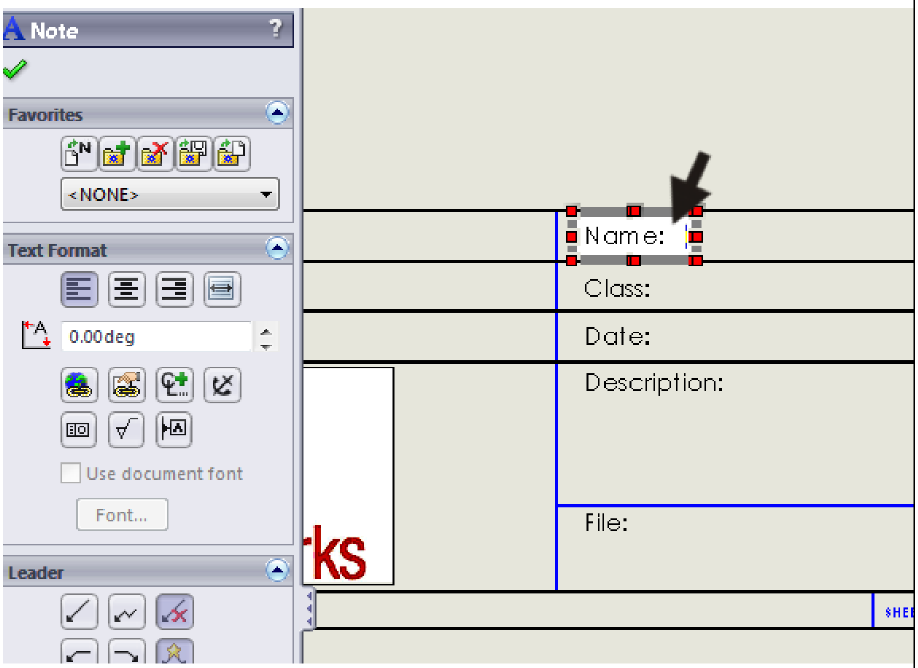
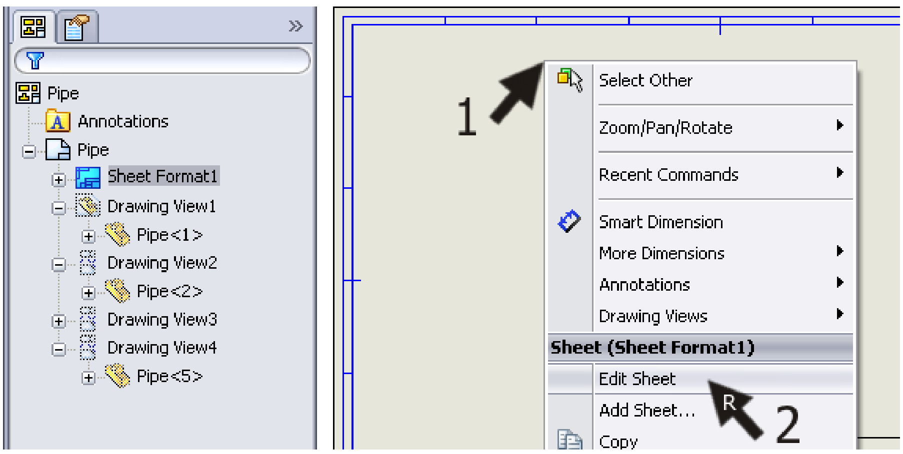
1. Select the size in the drawing.
2. Replace the text ‘<MOD-DIAM>’ with the capital ‘M’ under ‘Dimension Text’ in the PropertyManager. The field text will read: ‘M<DIM>’.
3. Click on OK.
</td>
<td>


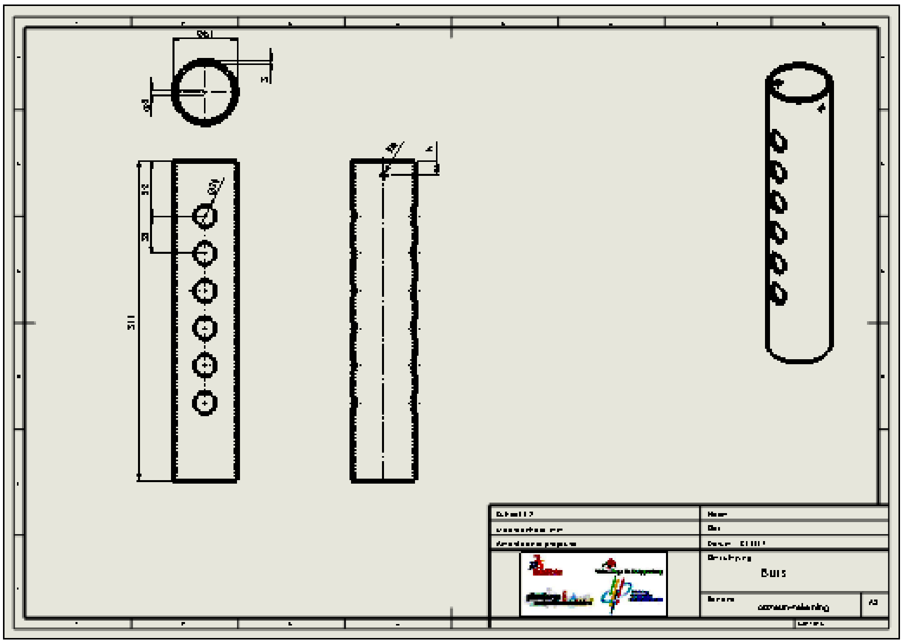
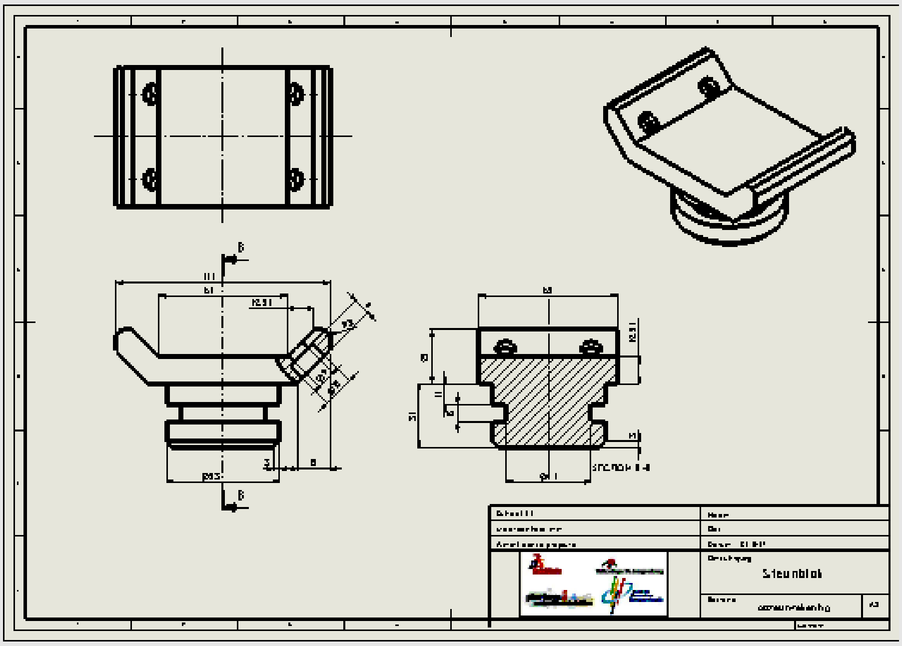
</td>
</tr>
<tr>
<td>25</td>
<td>
Finally, we add a centerline to the right-side view.

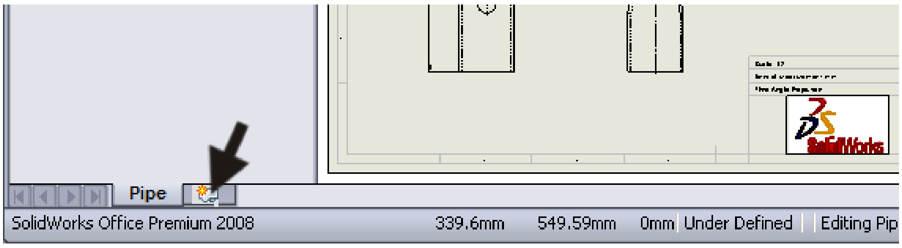
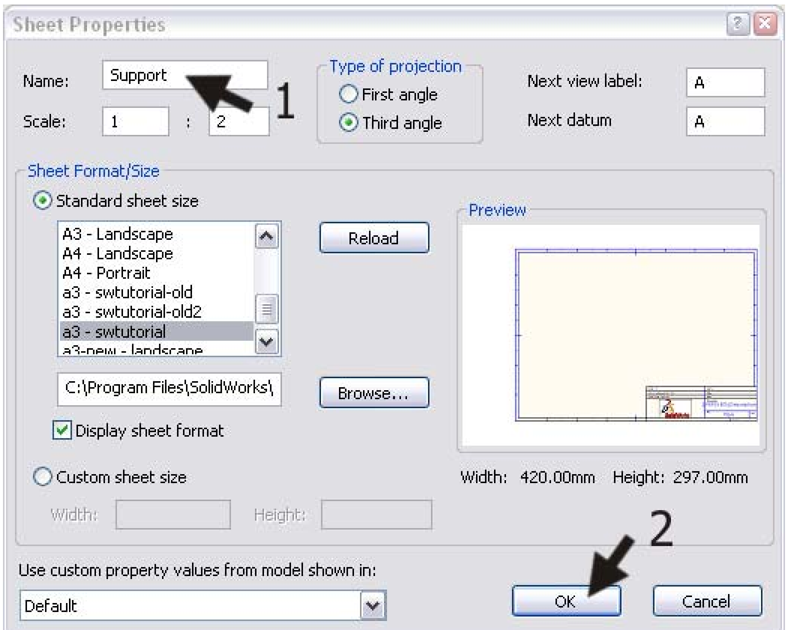
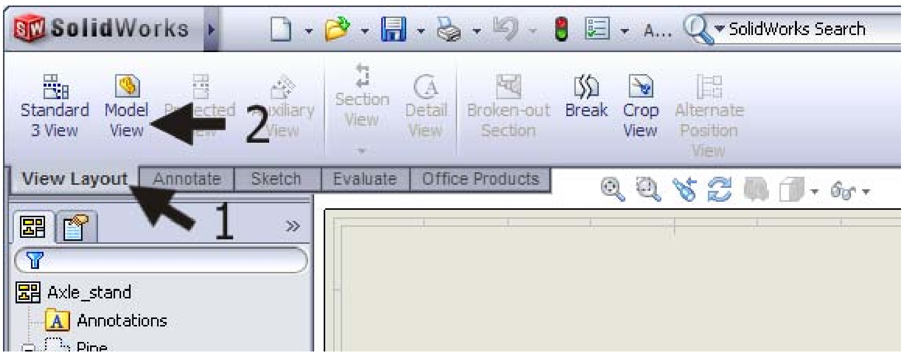
1. Select the tube with a click.
2. Click on ‘Centerline’.

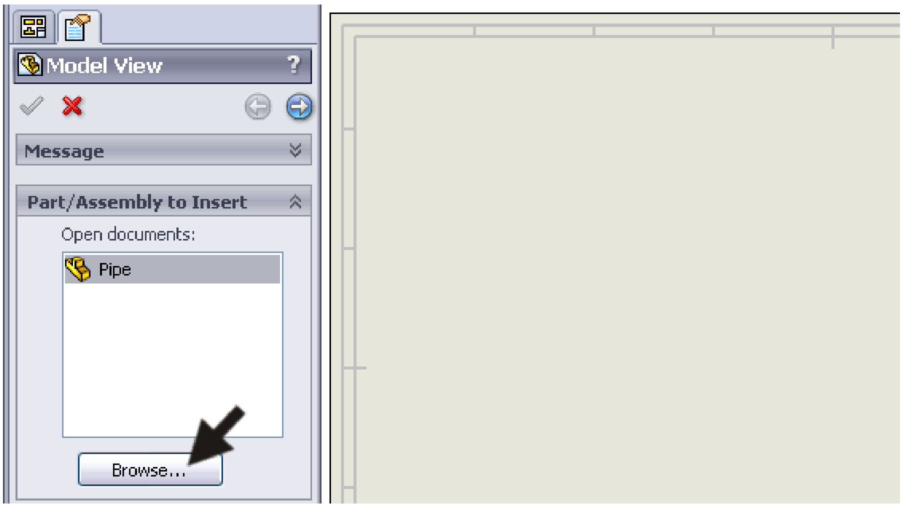
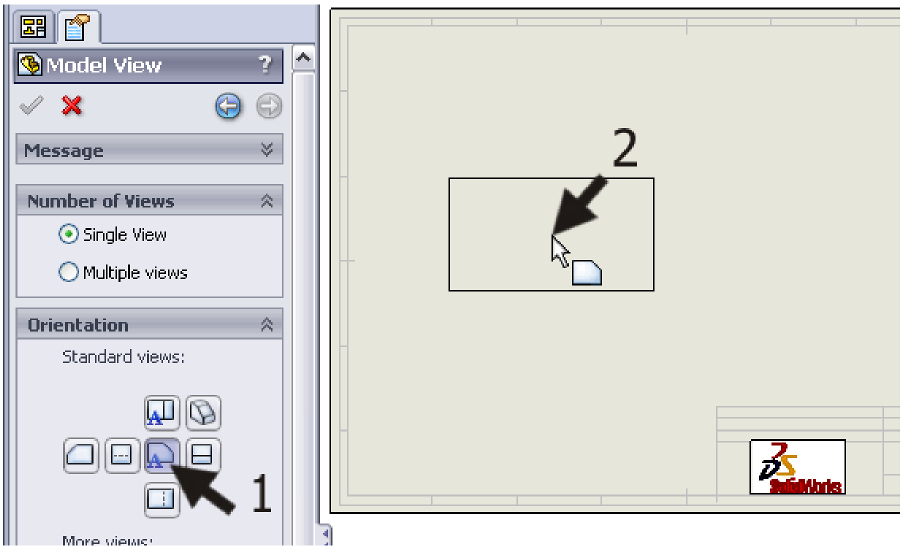
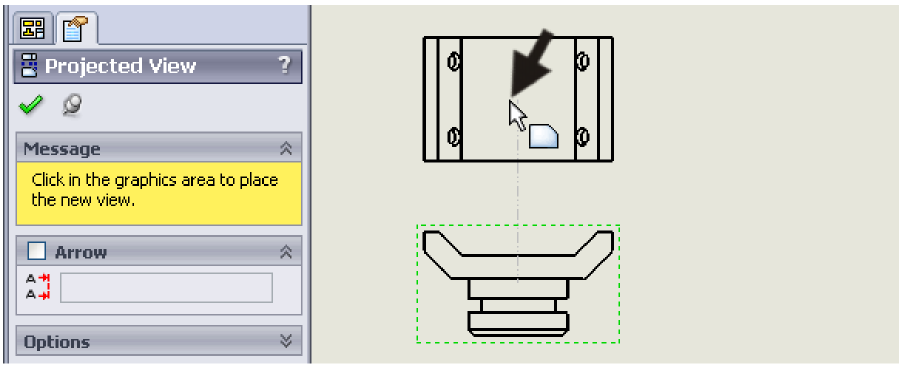
Try to click on another view as well and push the <Esc>key to end the Centerline command.
</td>
<td>


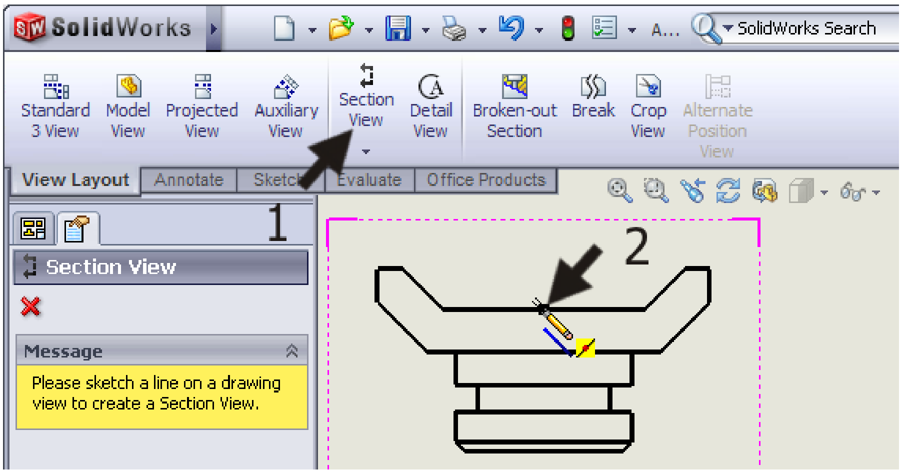
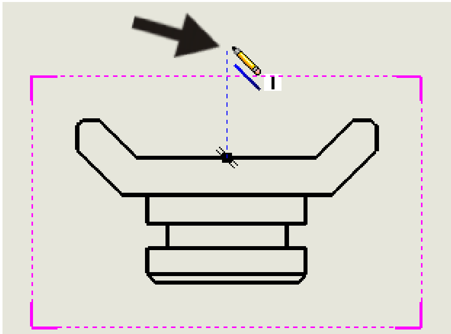
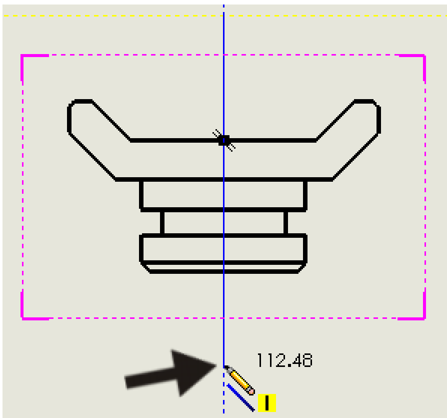
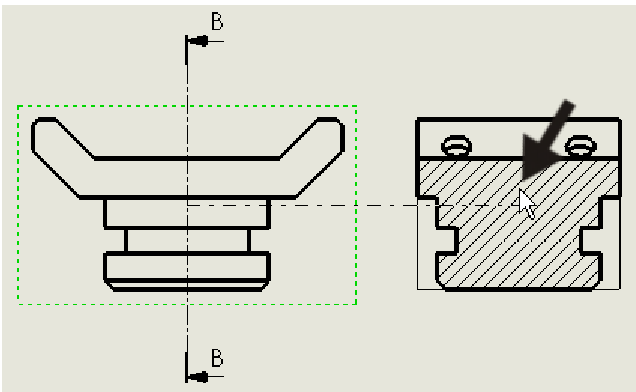
</td>
</tr>
</table>

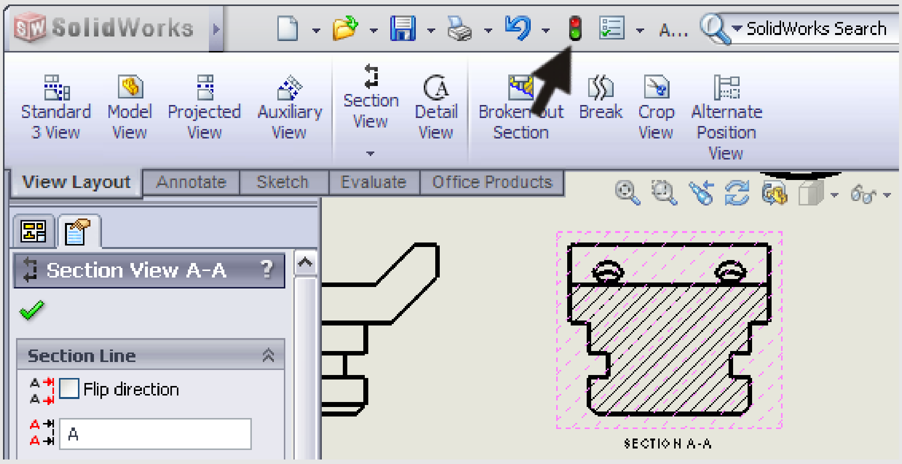
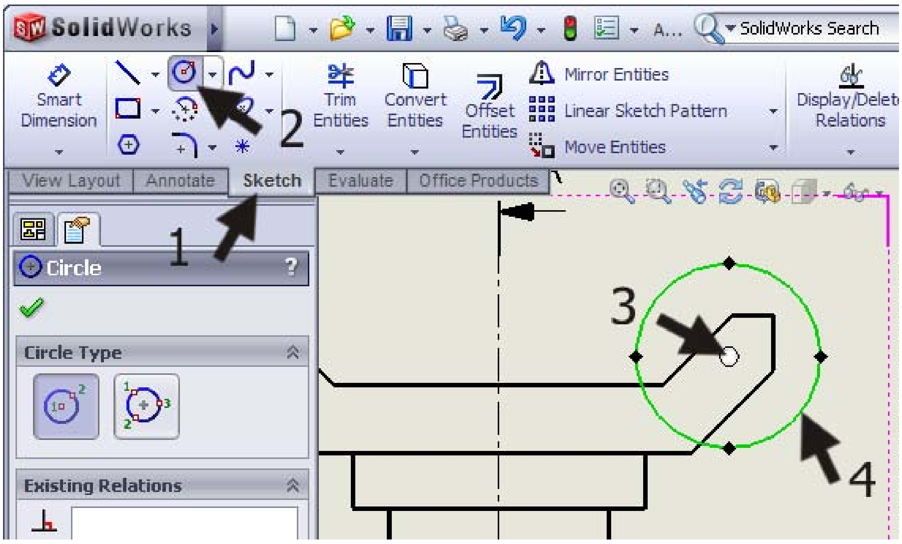
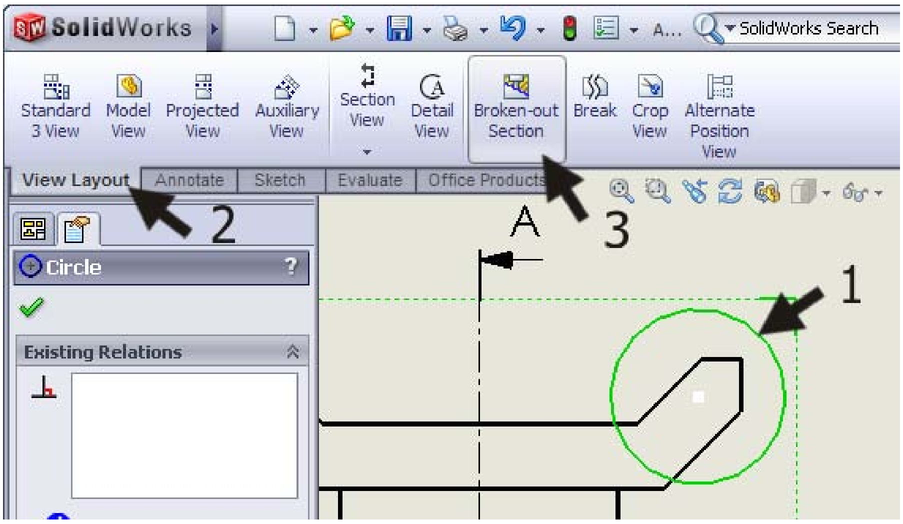
26	The drawing is now finished. You have to fill in your name in the right bottom corner, in the title block. 1. Right-click at a random position on the sheet (not on a view or a dimension). 2. Click on 'Edit Sheet Format' in the menu. The drawing will disappear temporarily.	
27	Zoom in on the right bottom corner. Double-click on the text field 'Name' and fill in your own name. Do the same with class. The other text fields – such as Date, Description and File – will be filled in automatically by SolidWorks.	
28	Right-click somewhere on the page and select 'Edit Sheet' again.	

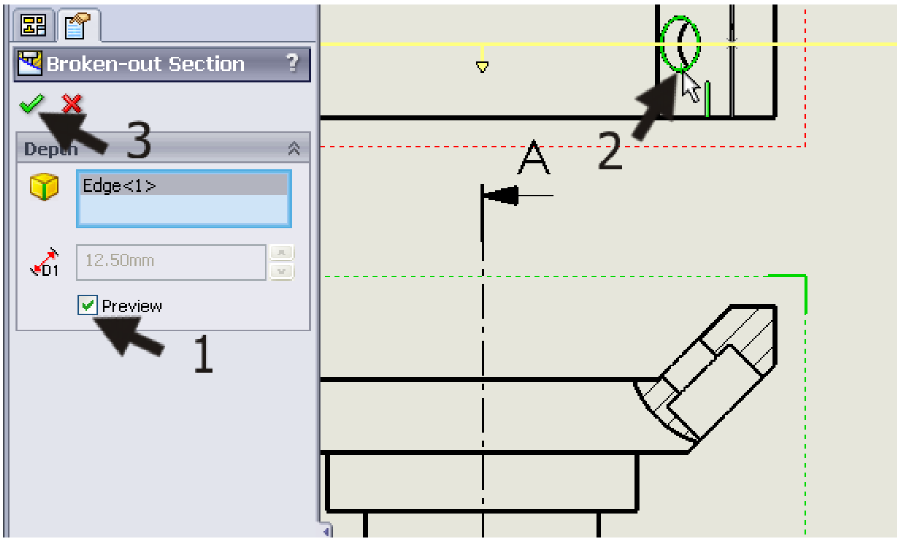
29	Save the file as: Axle_stand.SLDDRW.	
30	Print the drawing. You can find the most important settings for the printer commands in Tutorial 6. Ask your teacher for the right settings for the printer.	
	Work plan	Next we will make a drawing of the support block. In this drawing we will learn how to work with cross-cuts. You will also see how to change dimensions in a drawing. 

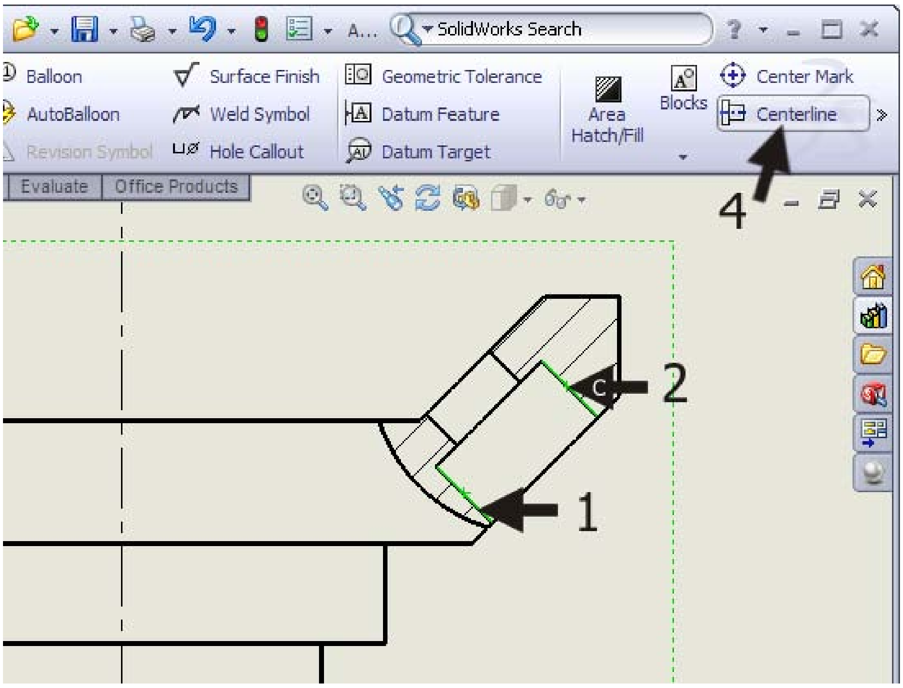
31	Add a new sheet to the file first: 1. Right-click on the tab at the bottom of the screen. 2. Select 'Add Sheet' in the pop-up menu. You have two tab sheets now; you can toggle between the drawings if you want to.	
32	Right-click somewhere on the new drawing sheet and select 'Properties...'. Name the sheet: 'Support'. Make sure the settings match those of the first sheet (Step 10).	
33	Click on 'View Layout' in the CommandManager, and then on 'Model View'.	

34	When the part 'Support' is opened, select it in the list in the PropertyManager. If not, click on 'Browse...' and find the file on your hard disk or on your memory stick.	
35	Click on the sheet to place the front view.	
36	The command 'Projected View' will start automatically now. Also set the top view and the isometric view of the support block.	

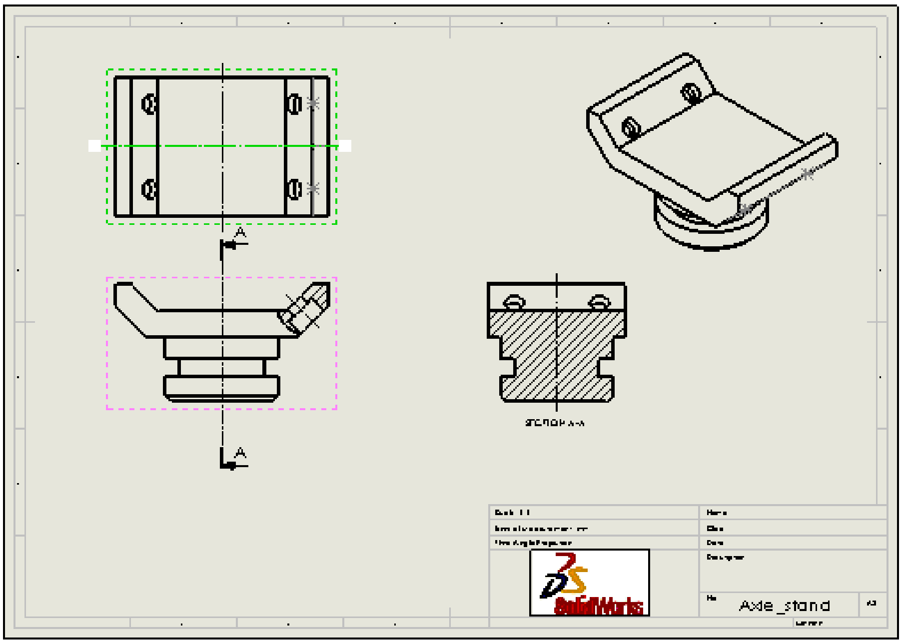
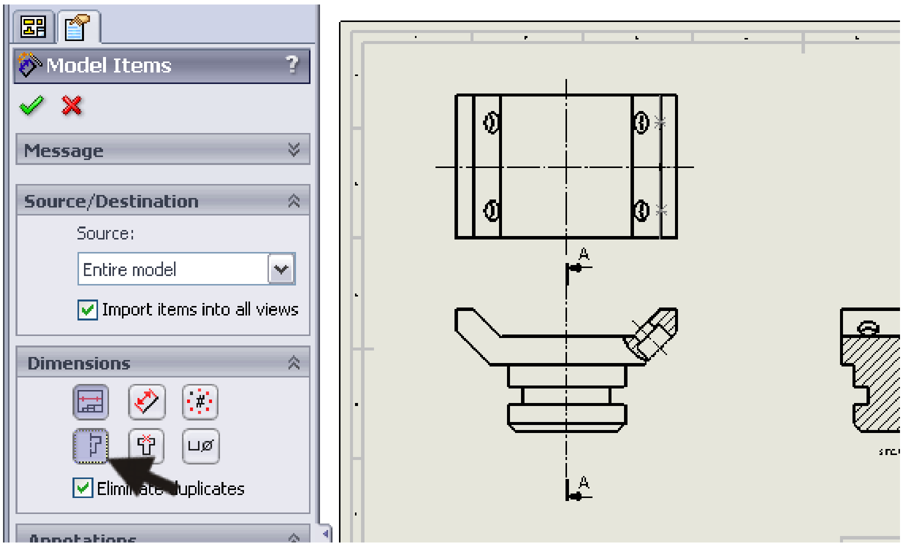
37	Let's make a cross-cut! 1. Click on 'Section View' in the CommandManager. Draw a cross-cut line. Be careful. Do this very accurately! 2. Set the cursor on top of the midpoint of the upper horizontal line, but **do NOT click!**	
38	Move the cursor upward; you can see a blue dotted line. Click above the view.	
39	Move the cursor downward now and click underneath the view.	
40	Next, the cross-cut will appear and you can place it beside the view.	
	Tip!	When you accidentally shift the cross-cut line, colored shading will appear

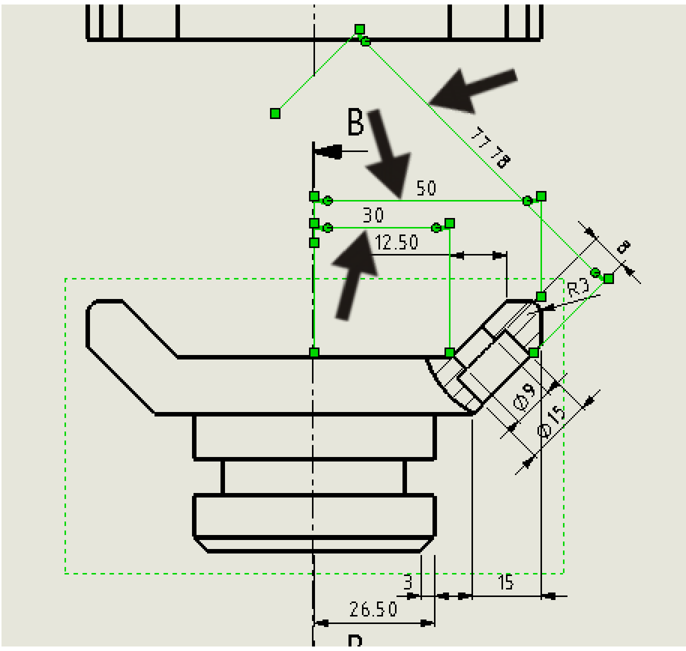
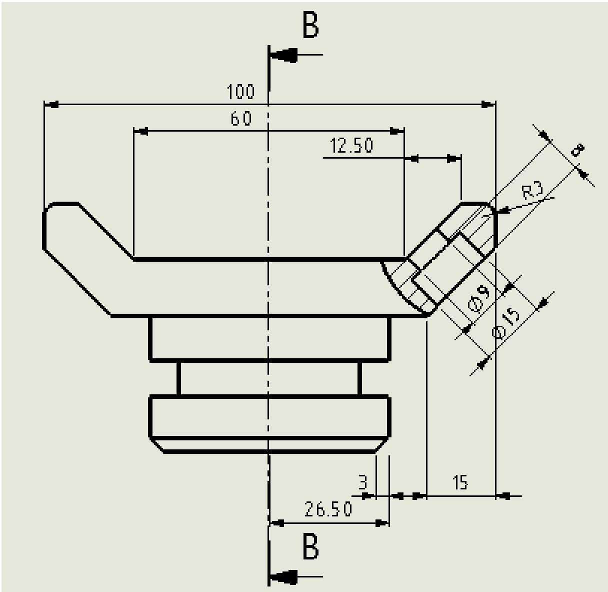
		in the cross-cut drawing. This indicates that the model has to be updated. In such a case, click on the Rebuild button in the standard toolbar. The colored hatching will disappear. 
41	To get a better view of the countersink hole, we will open a part of the front view. Click on 'Sketch' and after that on Circle in the CommandManager. Set the circle just about the same as in the illustration on the right.	
42	1. Make sure the circle is selected (it turns green). 2. Click on 'View Layout' in the CommandManager. 3. Click on 'Broken-out Section'.	

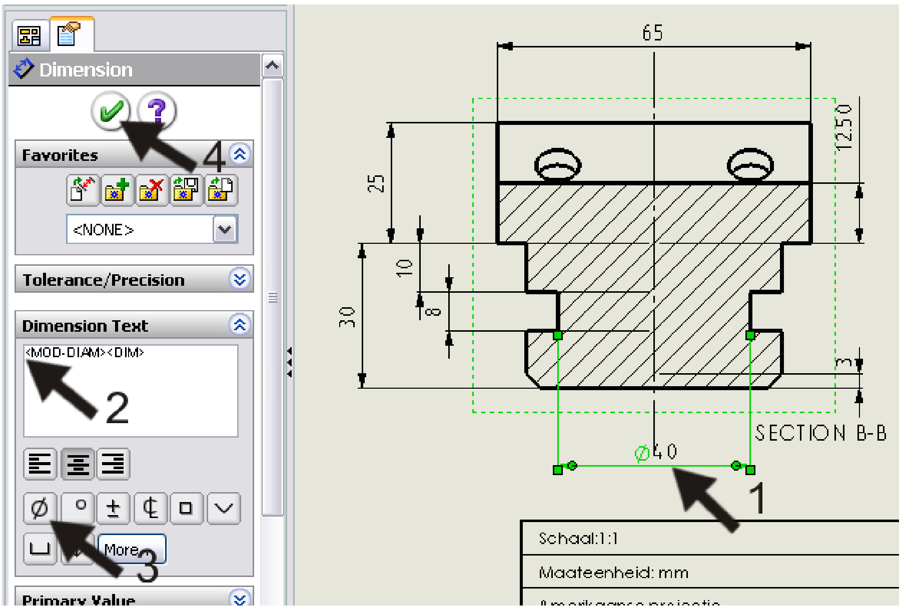
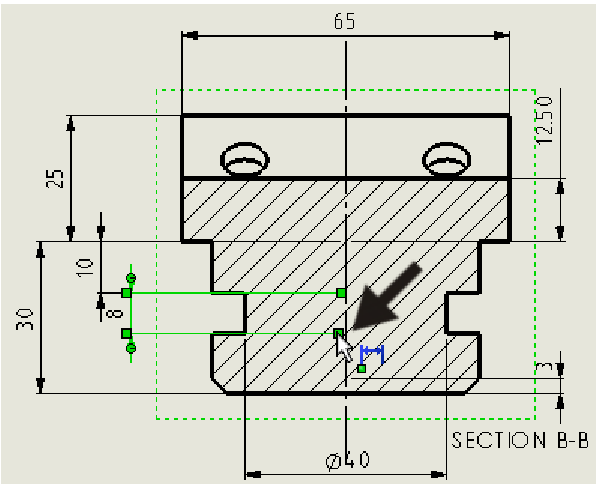
<table>
<tr>
<td>43</td>
<td>
1. Check the option ‘Preview’ in the PropertyManager.
2. Click on the edge of the hole in the top view. The cross-cut will run through here.
3. If the preview looks good, click on OK.
</td>
<td>


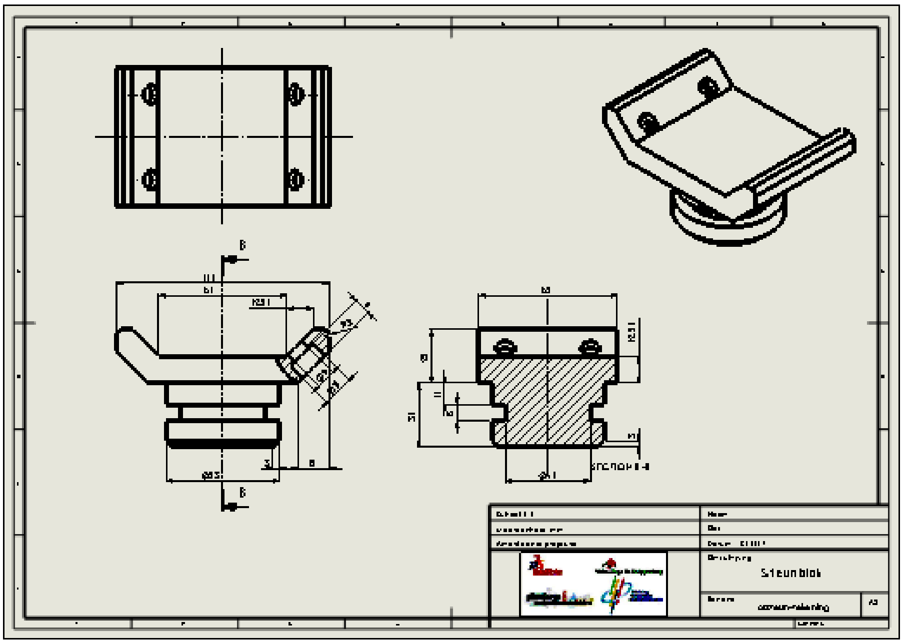
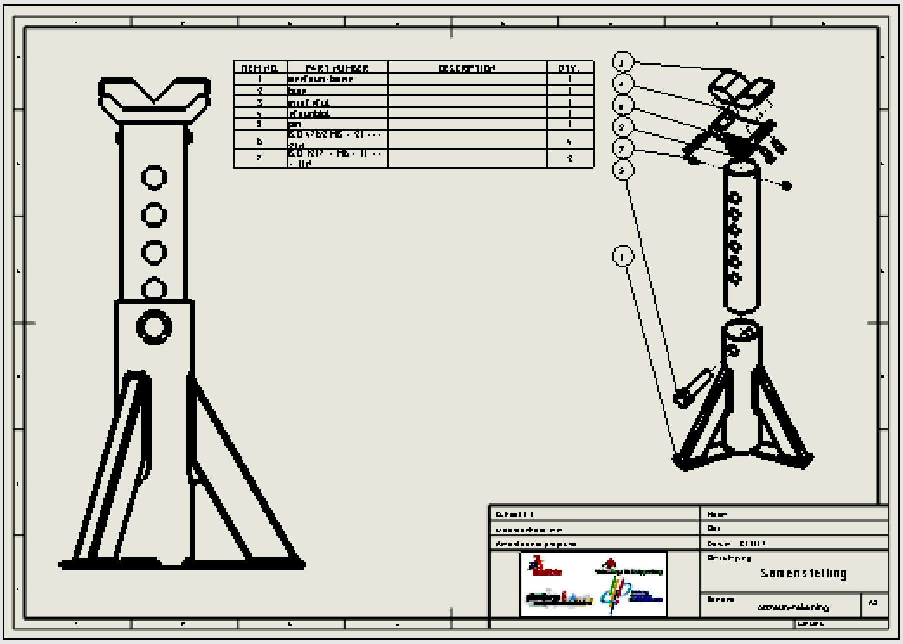
</td>
</tr>
<tr>
<td>44</td>
<td>
To put a centerline in the hole use the following steps:
1. Select the first edge from the hole.
2. Hold the <Ctrl> key and select the second edge from the hole.
3. Click on the tab ‘Annotate’ in the CommandManager.
4. Click on ‘Centerline’.
The centerline is a bit short now, but you can drag the end to extend it.
</td>
<td>


</td>
</tr>
</table>

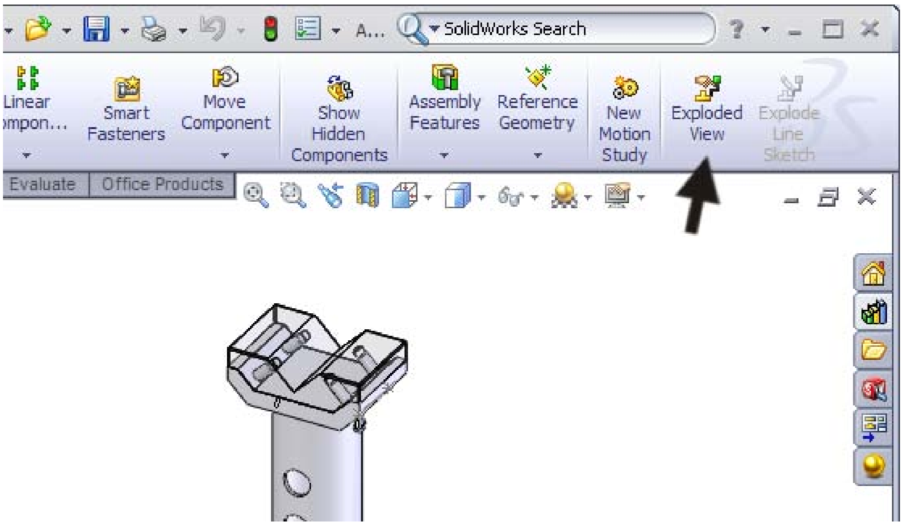
45	Add the other centerlines too, so the drawing will end up looking like the illustration on the right.	
46	Set the dimensions in the drawing now. Click on 'Annotate' in the CommandManager and then on 'Model Items'. Use the same setting as in the last drawing (Step 14). Make sure that the option Hole Wizard Profiles is also checked. Click on OK.	

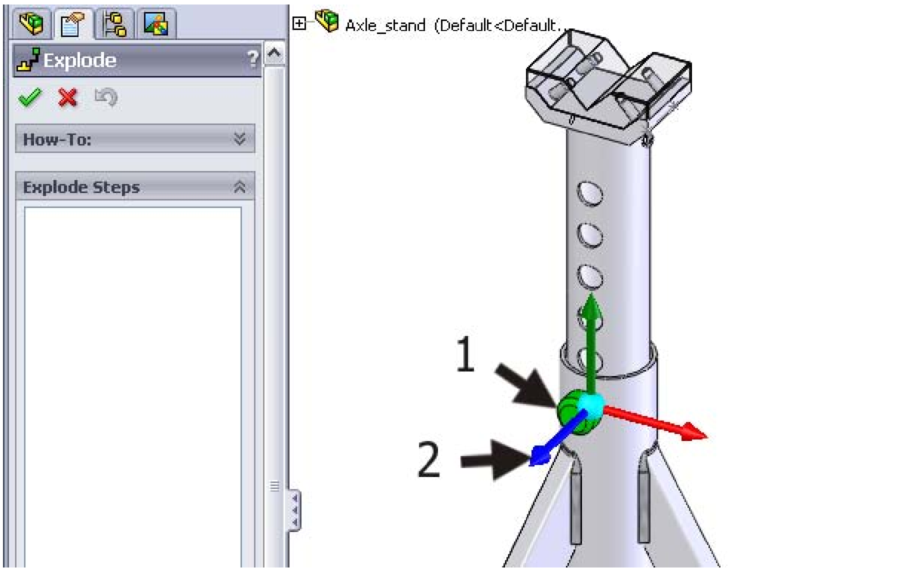
47	Shift the dimensions where necessary so your drawing looks orderly. Select the three dimensions as shown on the right. Push the <Del> delete key on the keyboard to remove them.	
48	Next, set two new dimensions. Click on 'Smart Dimension' in the CommandManager and set the dimensions of '60' and '100' like in the illustration. You have also used 'Smart Dimension' in sketches so you should be familiar with this command.	

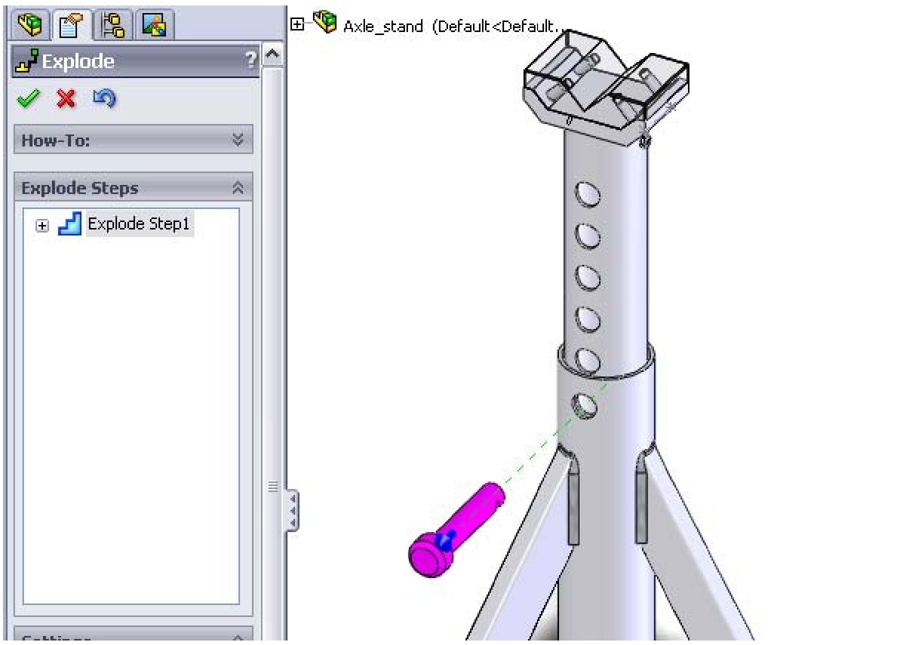
49	Replace the size R20 with R40 in the same way. Follow the next few steps to set a Ø symbol in front of the actual size: 1. Select the dimension. 2. Set the cursor at 'Dimension Text' in front of the existing text '<DIM>' in the PropertyManager. 3. Click on the diameter-symbol. In the text field it reads: '<MOD-DIAM><DIM>'. **4.** Click on OK.	
50	At some point you will see the auxiliary lines from the dimensions running through the view. You can easily drag the endpoint of the lines to the outside of the view or cross-cut.	
	Tip!	Notice that we have inserted dimensions in the drawing in two different ways: 1. By importing them from the 3D model. 2. By putting them in the drawing manually with the Smart Dimension command. There is an important difference between the two kinds of dimensions. When you double-click on an imported size, you will get a small menu in which you can change it. When you do so, the 3D model will also change! So be careful with this function. They are also called Driving Dimensions.  It is not possible to change a manually placed dimension. If you double-click on those, nothing happens. These are Driven Dimensions.

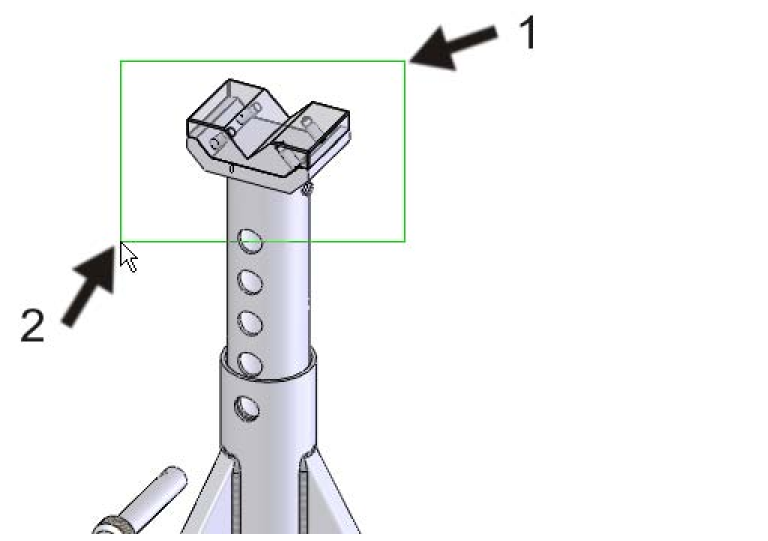
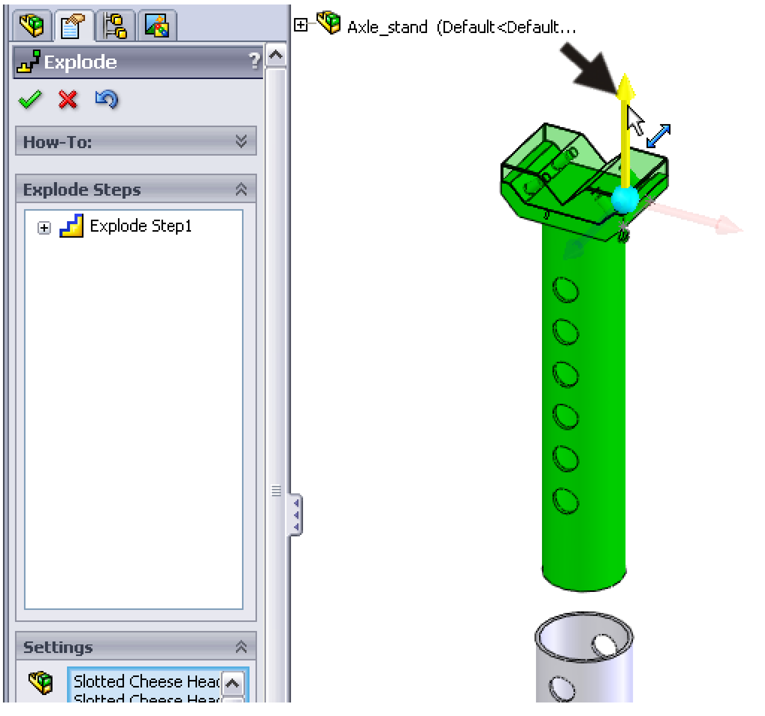
51	Fill in the right bottom corner of the drawing as you have done before (in Steps 26 to 28). Click on Save to save the file.	
	Work plan	Finally, we will make the drawing of the assembly. For this we use one of the views and an exploded view. To put an exploded view in the drawing, we have to make it in the assembly first. 

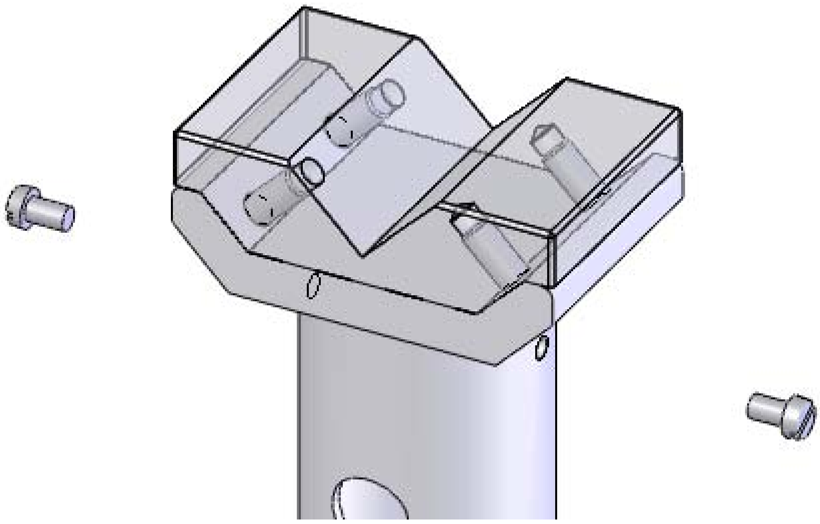
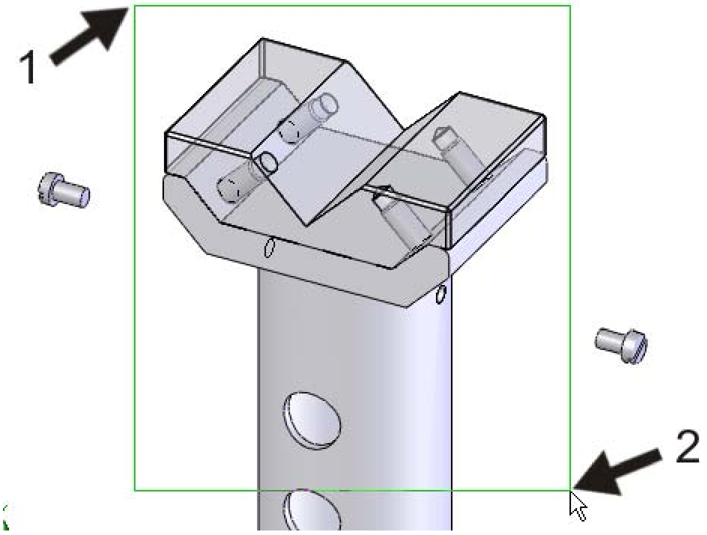
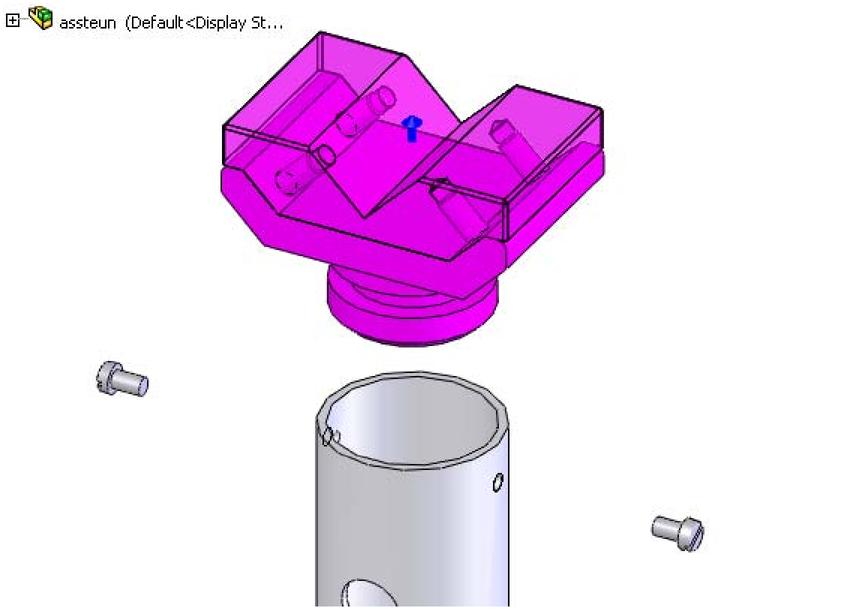
<table>
<tr>
<td>52</td>
<td>Open the file Axle_stand.SLDASM.

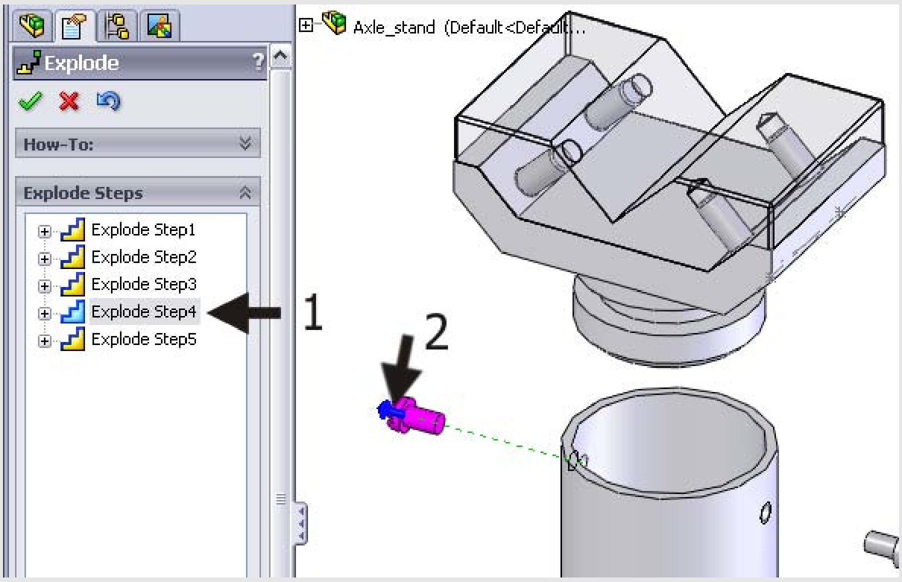
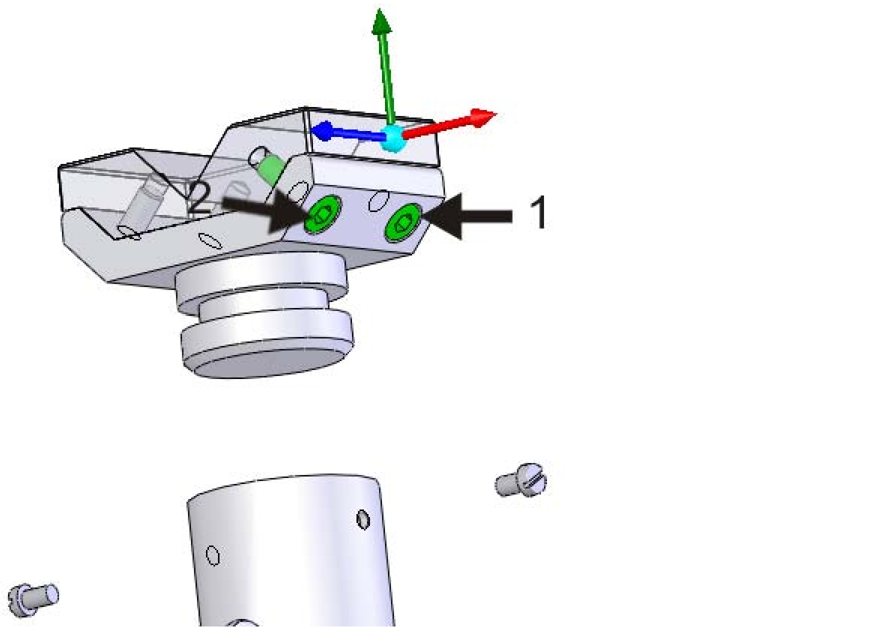
Click on ‘Exploded View’ in the CommandManager.</td>
<td>
</td>
</tr>
<tr>
<td>53</td>
<td>Click on the pin.

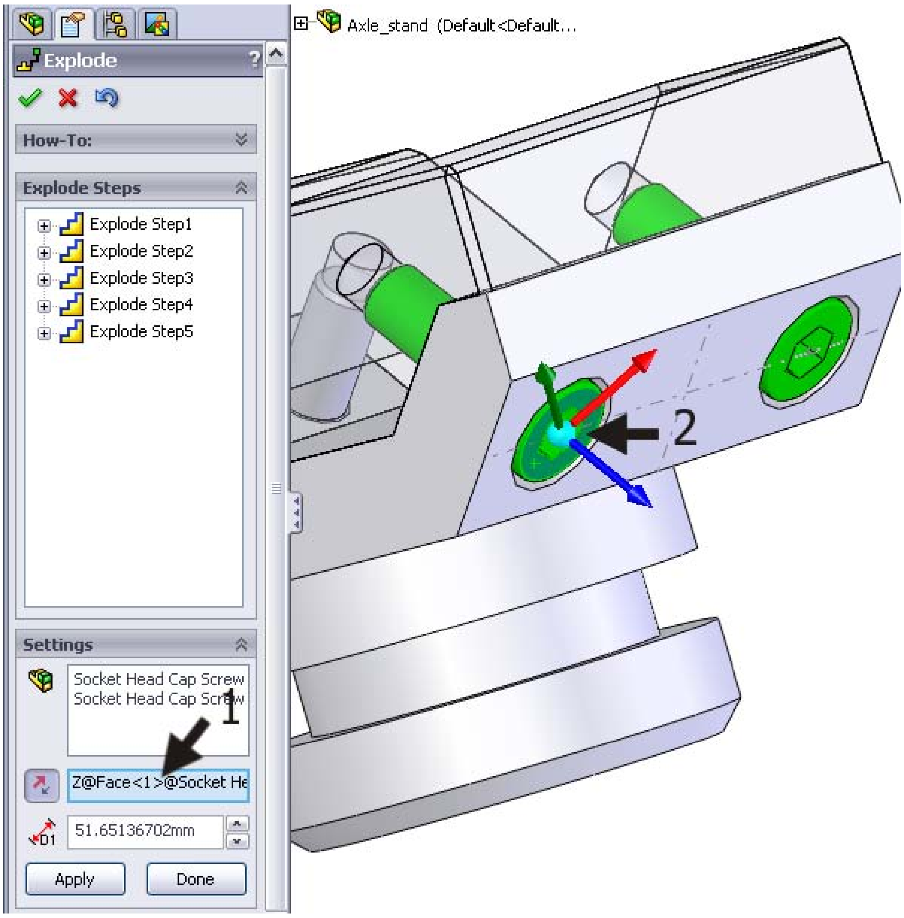
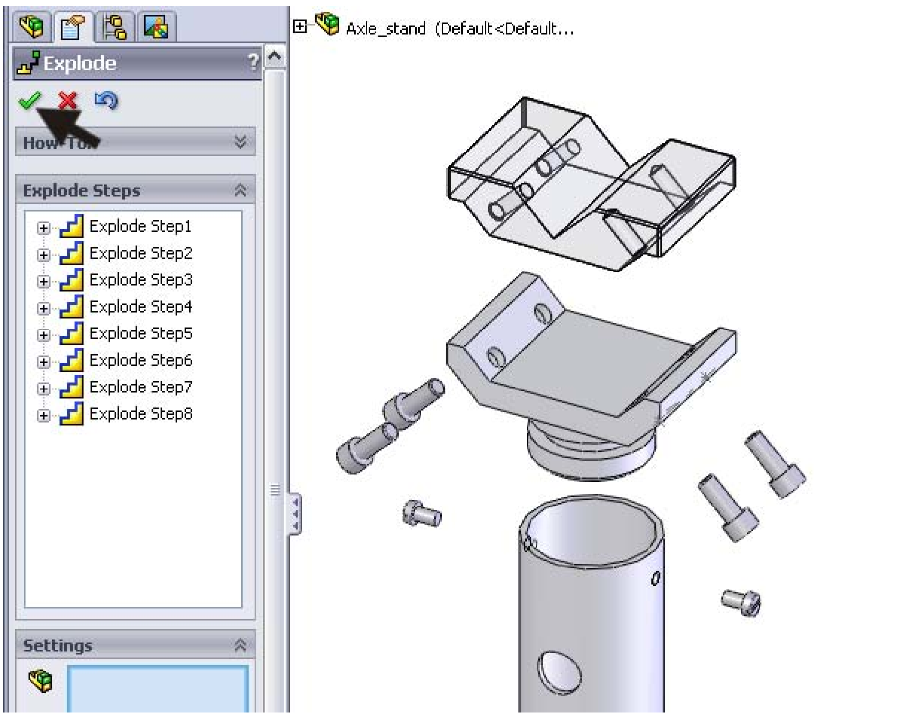
At the pin three arrows appear and you can now drag this part in three different directions.</td>
<td>
</td>
</tr>
<tr>
<td>54</td>
<td>Drag the part using the blue arrow, so the pin will end up beside the assembly.

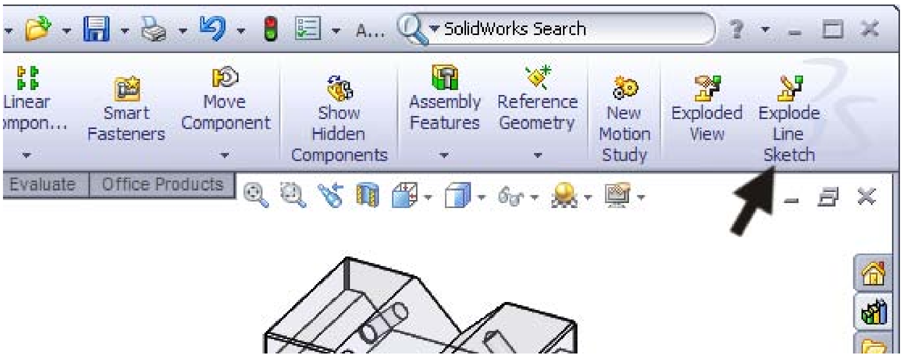
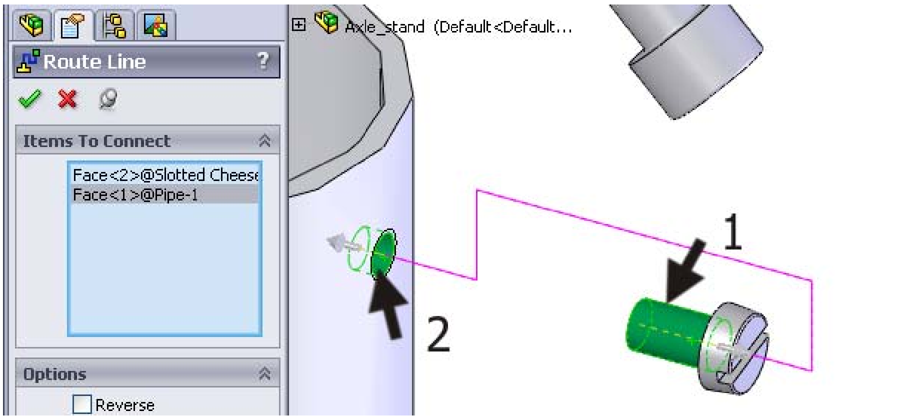
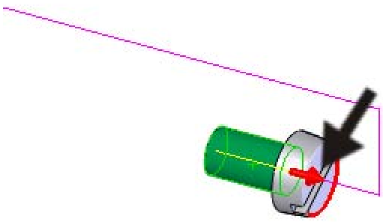
Click somewhere beside the model to un-select the pin.</td>
<td>
</td>
</tr>
</table>

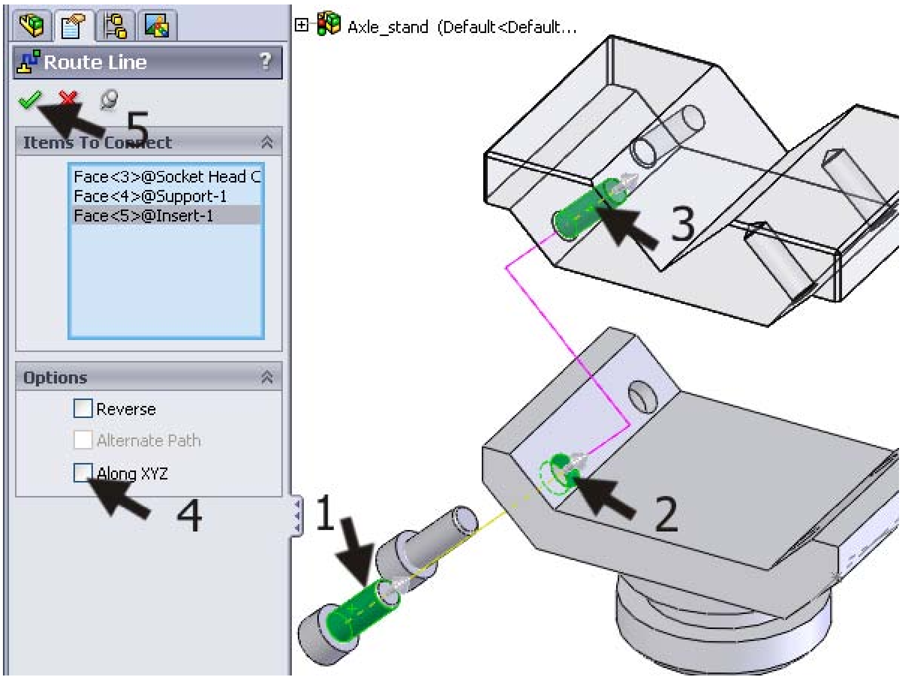
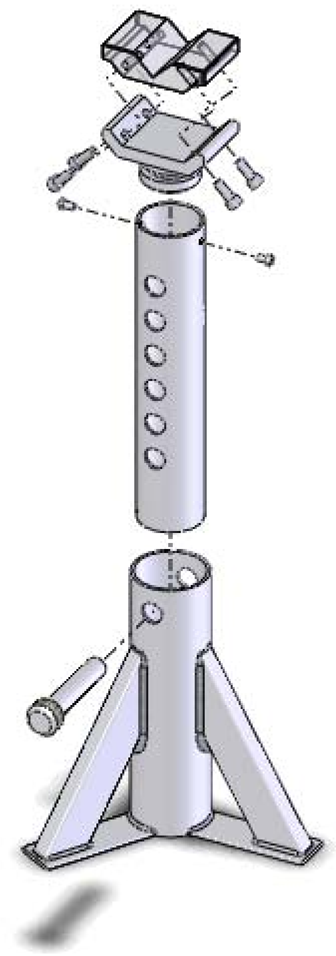
55	Drag a frame around the top of the support to select all parts of it. Drag from the **right** to the **left**, so the tube will be selected as well!	
56	Next, drag all selected items upward using the vertical arrow. Make sure the inner tube will extend above the base tube.	

57	Drag the two little screws just below the support block one by one until they are just outside of the model. You can easily rotate the model during this operation, but remember to put it back into the trimetric position. This is the only way to you will get a clear idea about how the drawing will look like at the end.	
58	Drag a new frame around the top, but drag it from left to right this time. The tube is not selected now. Make sure the support block is completely in the selection frame, including the ‘invisible’ part that is inside the tube.	
59	Drag the selected parts upward again.	

	Tip!	If the part is not directly in the right spot, you can always click on it a second time and drag it to a new position. However, this will create a new 'Explode Step' and make your model more complex. It is much better to find the step you want to change in the PropertyManager and then click on it. A blue arrow will appear on the part, and you can change its position by dragging the blue arrow. 
60	Rotate the model a little so you can see two of the screws in the bottom of the support block. Select the two screws.	

61	In order to be able to shift the screws in the right direction (at the same angle as the screw holes) follow the next few steps: 1. Click on the field Explode Direction in the PropertyManager. 2. Click on a plane from the screw. The arrows now change direction, and you can shift the screws in the right direction.	
62	Move the other two screws in the same way. Raise the insert. The parts are all in the right position now. Click on OK in the PropertyManager.	

63	To get a better idea about how the parts of the product fit together, the parts are often connected with lines. Click on 'Explode Line Sketch' in the CommandManager to do so.	
64	Select the two planes as shown on the right.	
65	You can see that the line from the screw starts at the head of the screw and it should start at the other end. Is this the same in your drawing? Click on the gray arrow at the beginning of the line. The direction will reverse now. When the line is ok, click on OK in the PropertyManager.	

66	Make a similar line for one of the screws in the support block. Select the three planes as shown.	
67	Draw all connection lines in the exploded view in the same way.	

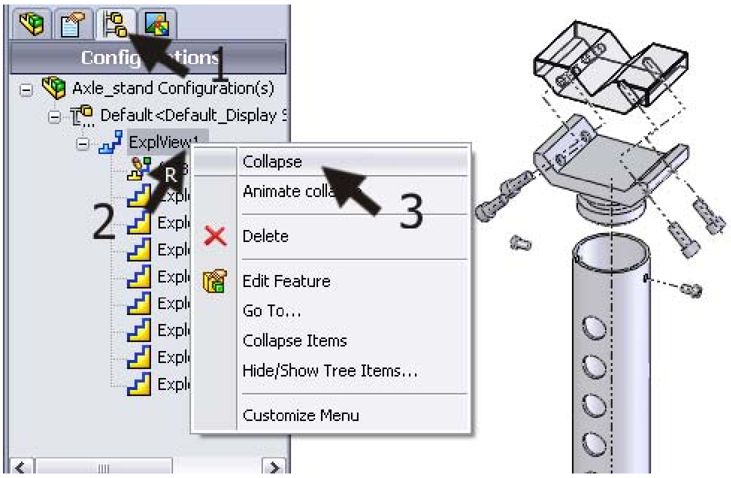
<table>
<tr><td>68</td><td>The assembly has now turned into an Exploded View. But how do we return to the normal assembly?

1. Go to the ConfigurationManager.

2. Right-click on ‘ExplView1’.

3. Select ‘Collapse’.

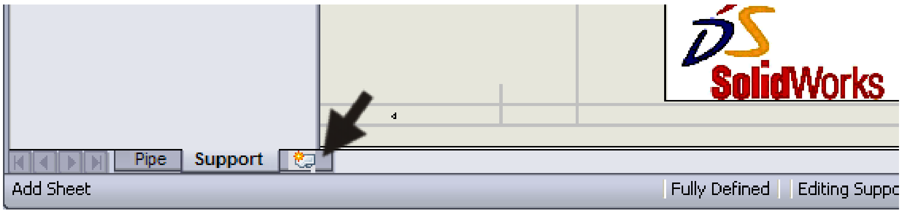
To return to the exploded view again, select ‘Explode’ in the same menu.

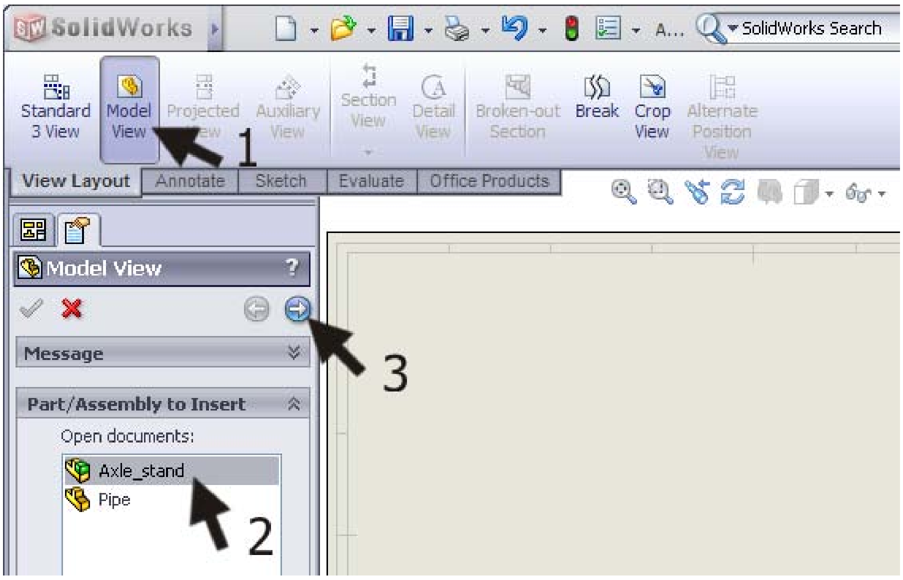
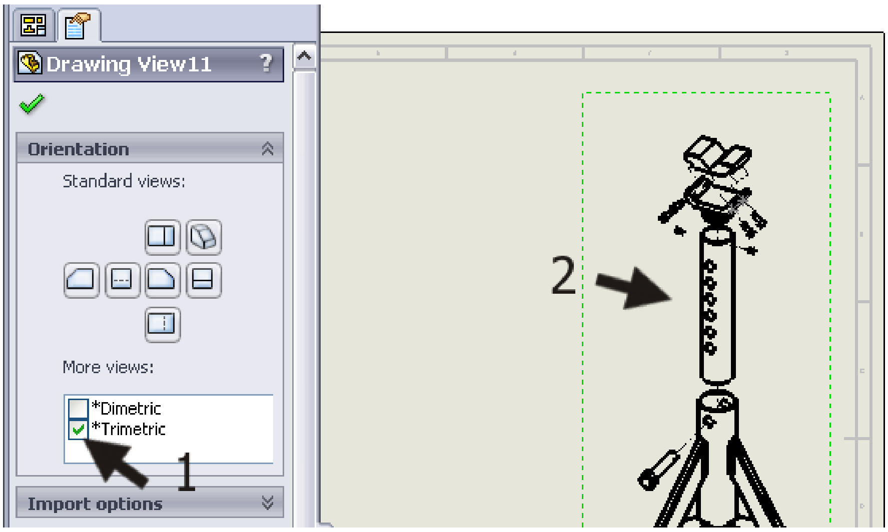
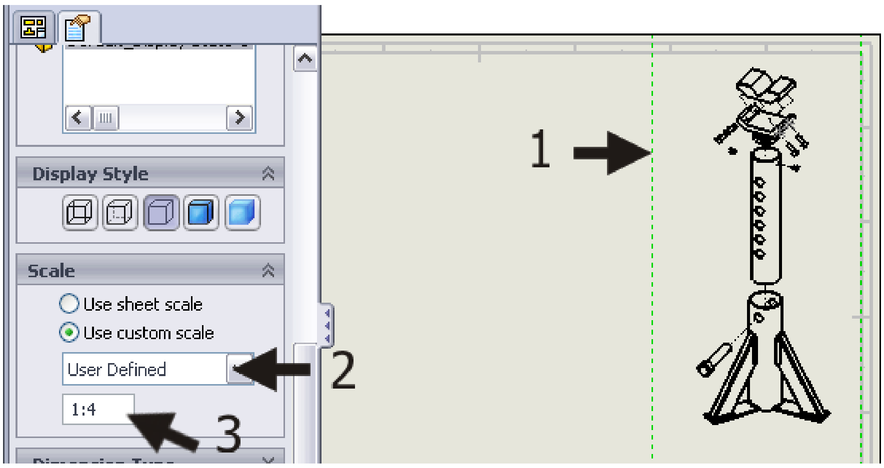
Try the option ‘Animate collapse/explode’. You will see the parts moving away from and toward each other.</td><td>
</td></tr>
<tr><td>69</td><td>Make sure the assembly is exploded and save the file.

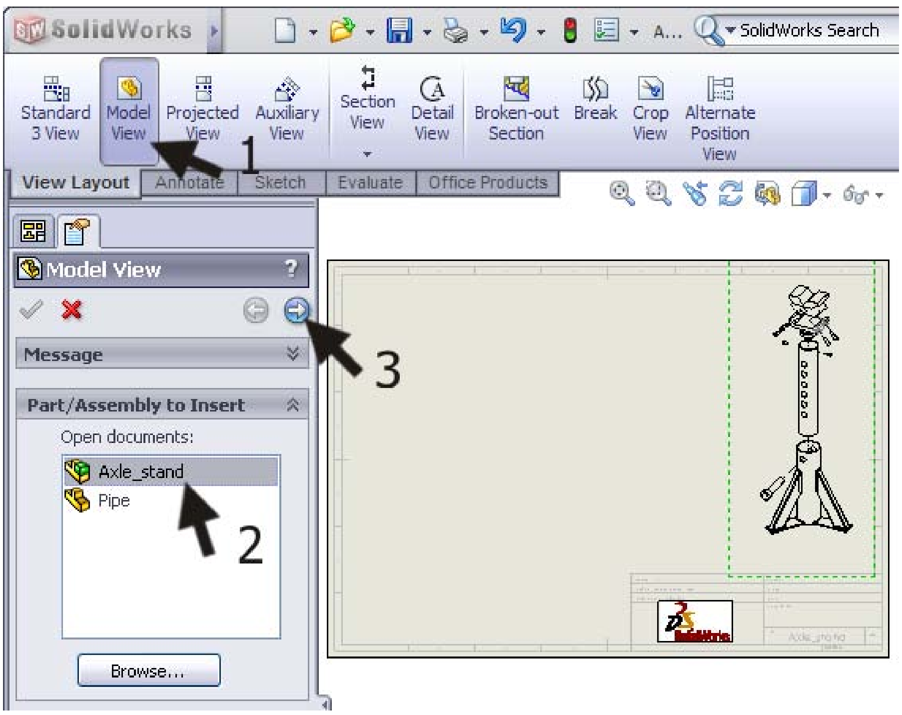
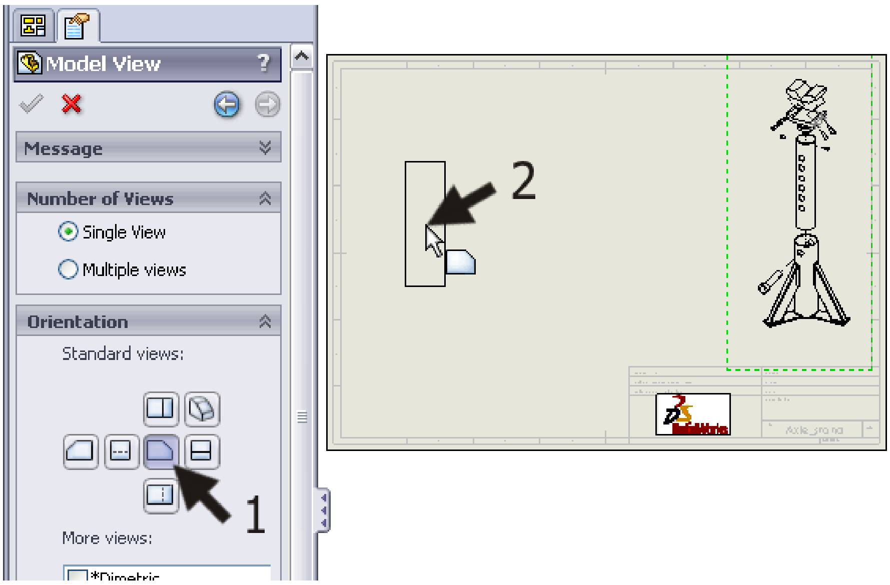
Return to the drawing again that you were working in.

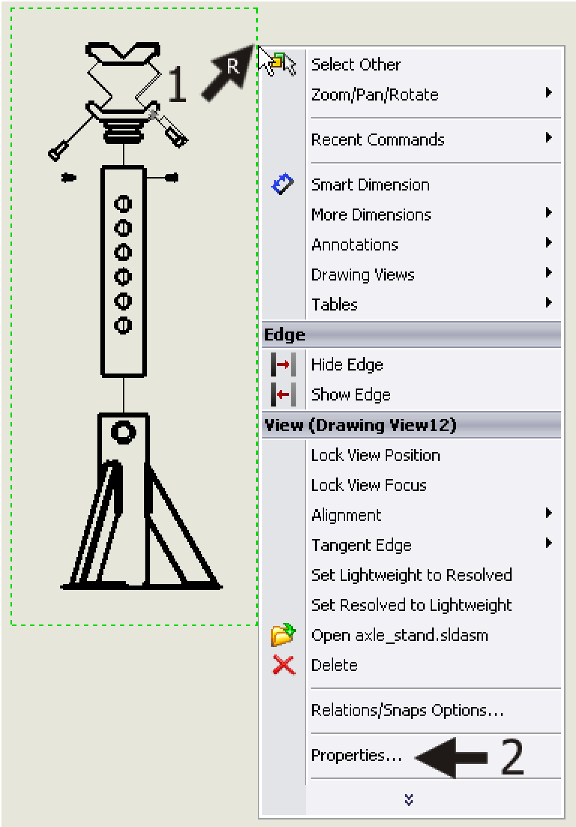
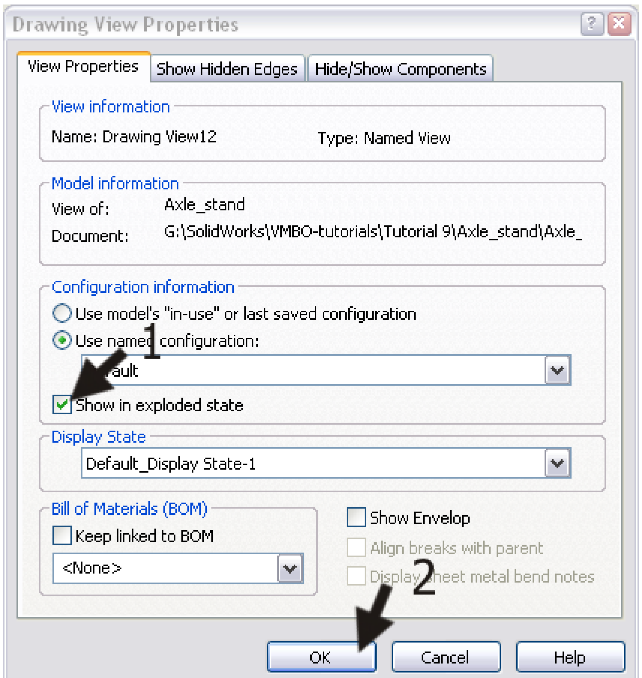
Push the capital ‘R’ on the keyboard.

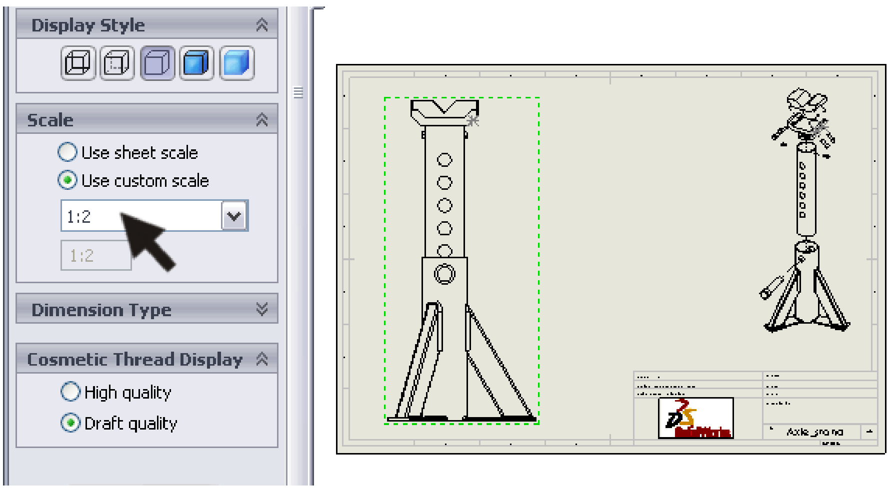
Click on Axle_stand.SLDDRW.</td><td>
</td></tr>
<tr><td>70</td><td>Add a drawing sheet to the file.

Click on Add Sheet.</td><td>
</td></tr>
</table>

71	1. Click on 'View Layout' in the CommandManager. 2. Select 'Model View'. 3. Select the file 'Axle_stand'. 4. Click on Next.	
72	1. Make sure the option 'Trimetric' is checked in the PropertyManager. 2. Put the view on the drawing sheet.	
73	The exploded view may be enlarged a little. 1. Select the exploded view. 2. Change the 'Scale' to 'User Defined' in the PropertyManager. 3. Set the 'Scale' to '1:4'.	

<table>
<tr>
<td>74</td>
<td>Next, we will put the front view on the sheet.
1. Click once more on ‘Model View’ in the CommandManager.
2. Select the file ‘Axle_stand’.
3. Click on Next.</td>
<td>
</td>
</tr>
<tr>
<td>75</td>
<td>1. Select the Front view in the PropertyManager.
2. Put the view on the drawing sheet.
Automatically, the command Projected View will start up. Click on OK to end this command.</td>
<td>
</td>
</tr>
</table>

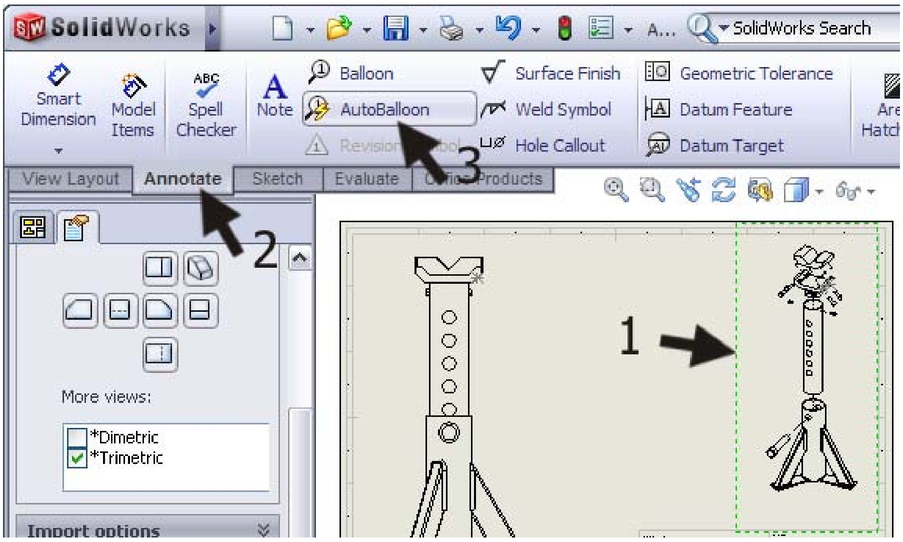
76	The front view is still exploded, but this is not meant to be. To change this, right-click on the View and select 'Properties…'.	
77	Uncheck the option 'Show in exploded state' in the menu that appears. Next, click on OK.	

<table>
<tr>
<td>78</td>
<td>Change the ‘Scale’ to ‘1:2’, as you have done in Step 73.</td>
<td>
</td>
</tr>
<tr>
<td>79</td>
<td>We will set the part numbers to the exploded view.

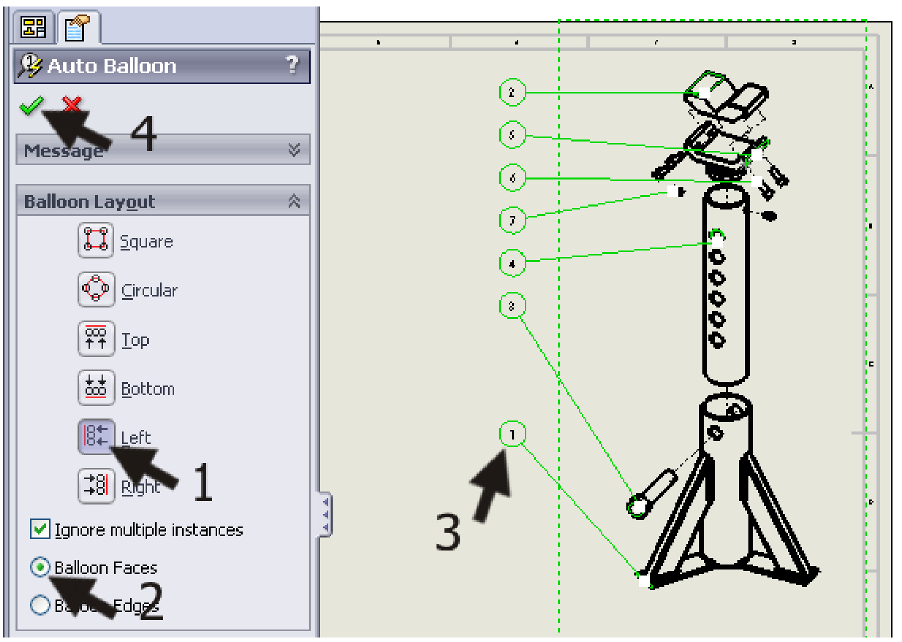
1. Select the exploded view.

2. Click on ‘Annotate’ in the CommandManager.

3. Click on ‘AutoBalloon’.

The part numbers now appear around the exploded view.</td>
<td>
</td>
</tr>
<tr>
<td>80</td>
<td>1. Click on the option ‘Left’ in the PropertyManager to set all part numbers at the left side of the exploded view.

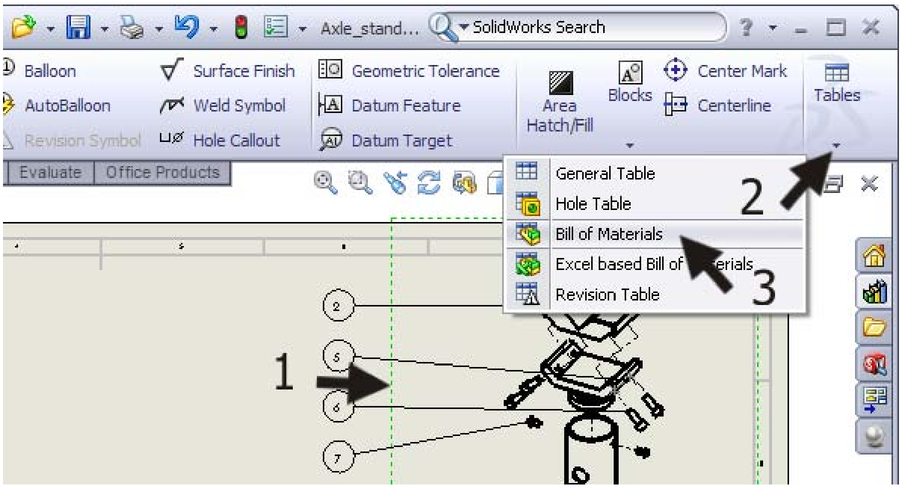
2. You can drag the numbers to put them on the right if you want to.

3. Click on OK.</td>
<td>
</td>
</tr>
</table>

81

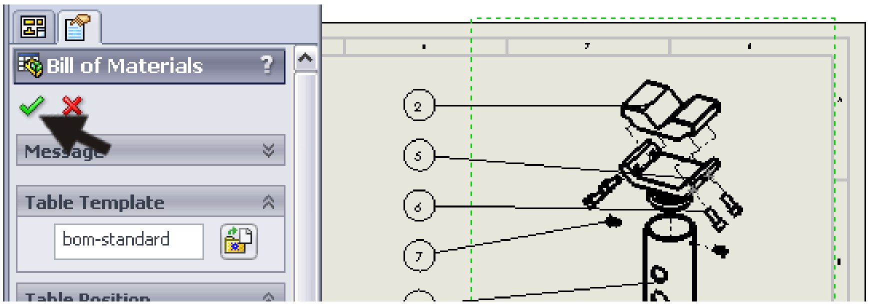
Finally, we will add a parts list.

1. Select the exploded view first.
2. Click on 'Tables' in the CommandManager.
3. Select 'Bill of Materials'.



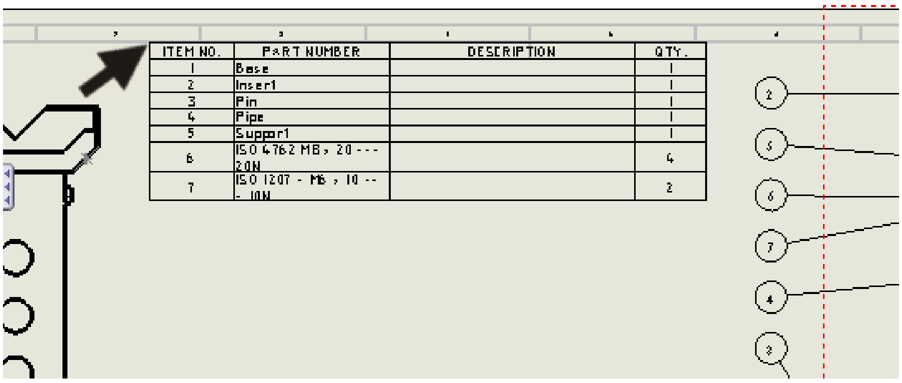
82

Click on OK in the PropertyManager.



83

Put the parts list on the drawing sheet and click OK in the PropertyManager.

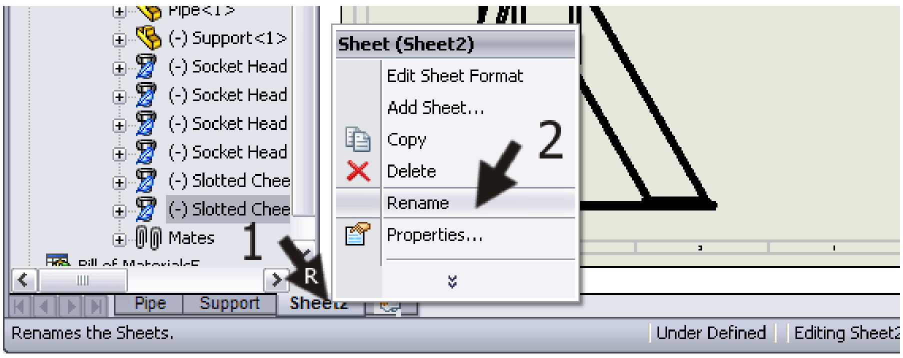


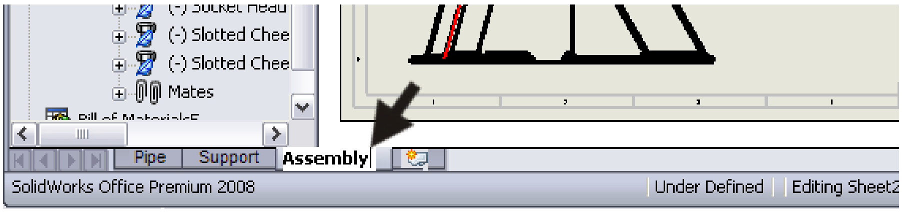
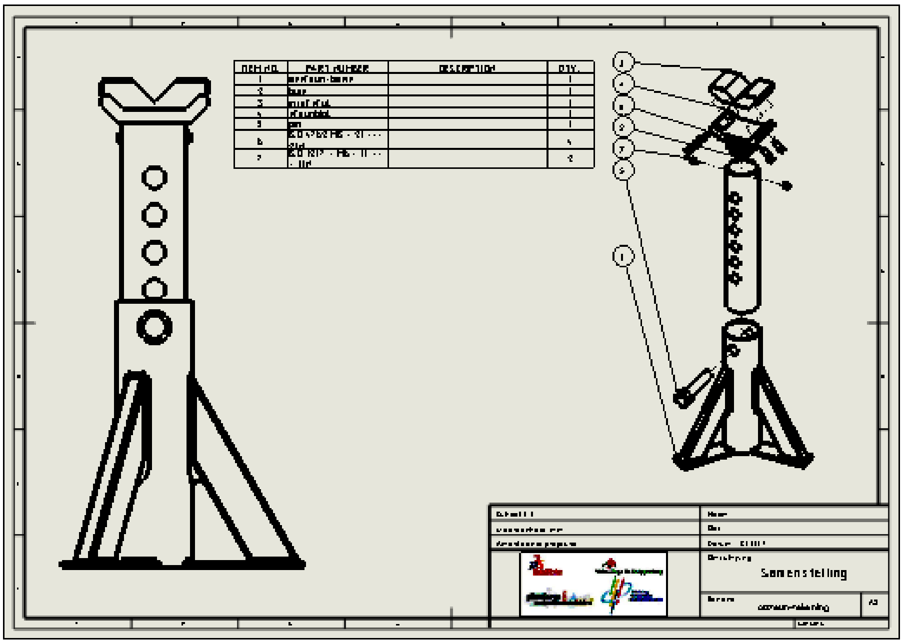
84

We must give our assembly drawing a name now. It is still called 'Sheet2' (or another number).

Right-click on the tab which contains the composition drawing.

Select the option 'Rename' in the menu.



85	Type in another name for the drawing, for example: 'Assembly'.	
86	Fill in your name in the right bottom corner.	
87	Save the drawing and print it.	
	What are the main features you have learned in this tutorial?	In this tutorial you have made three drawings and learned the most important functions for making a drawing. You have: - Placed views onto a drawing sheet. - Set dimensions in drawings, both automatically and manually. - Made some cross-cuts, including complete and partial cross-cuts. - Made an exploded view. - Added part numbers and a parts list to a composition drawing. There are a lot more functions to use when making drawing, but the things you know now will enable you to draw any products you want!

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

SolidWorks Benelux
RTC Building
Jan Ligthartstraat 1
1800 GH Alkmaar, Netherlands
Tel: +31 (0)72 514 3550
1800 GH Alkmaar, Netherlands
Tel: +31 (0)72 514 3550