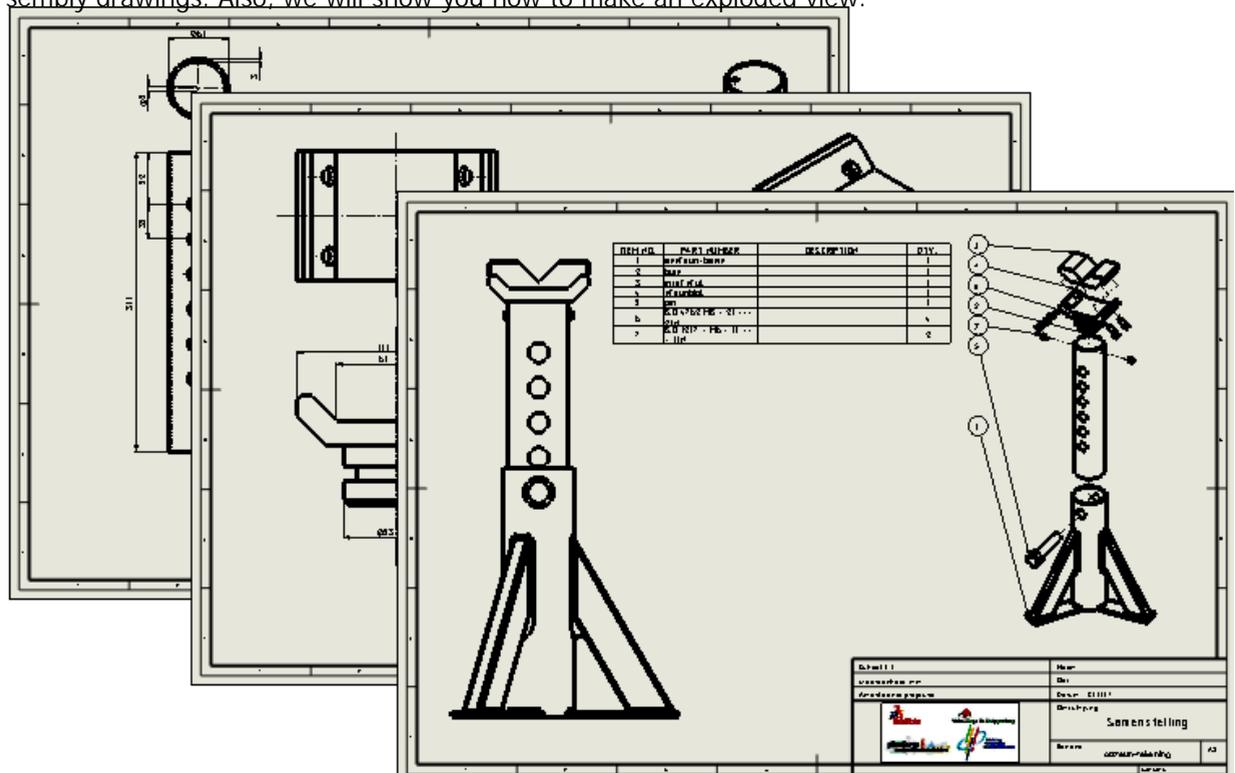




## Axle Support, the Technical Drawing

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.

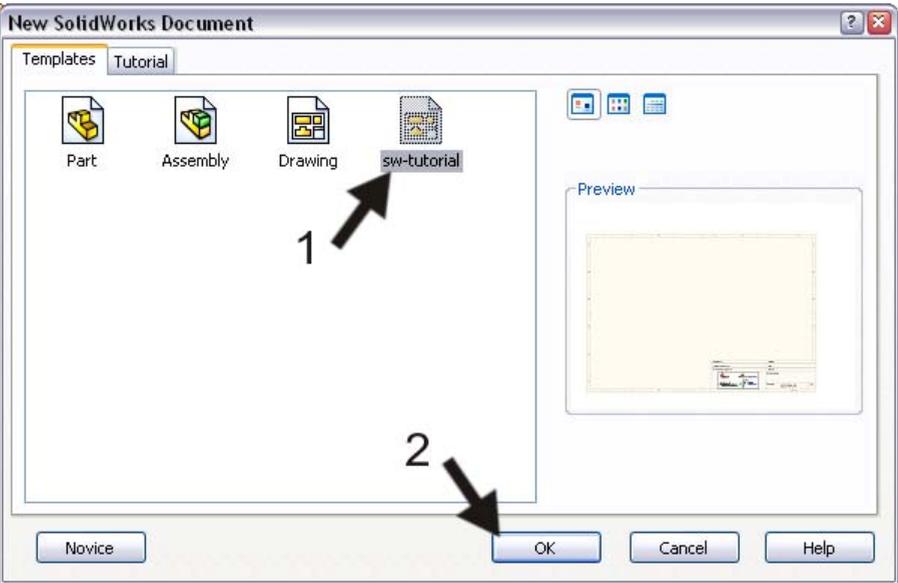
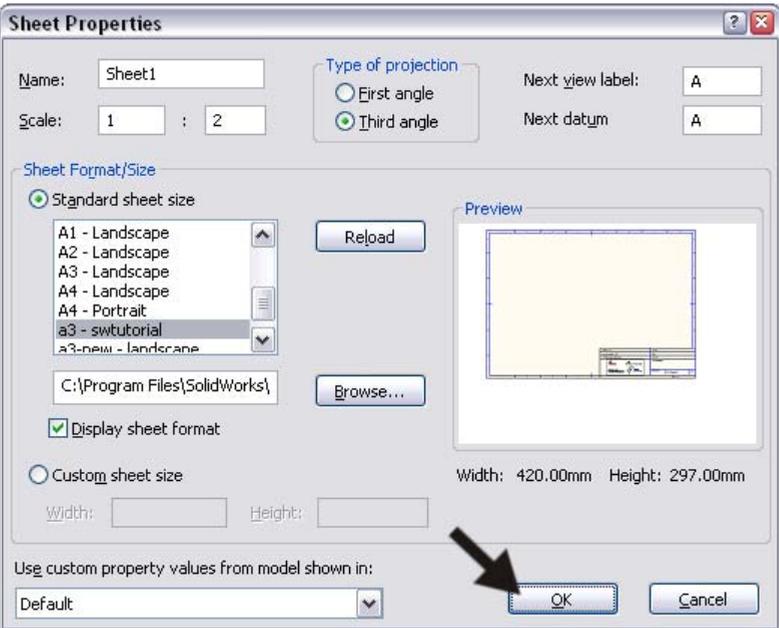
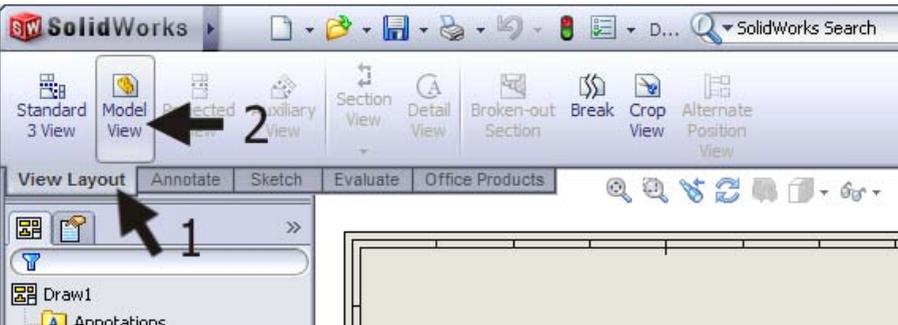
Creating a 2D drawing is not very difficult. We will show you a number of examples of single part and assembly drawings. Also, we will show you how to make an exploded view.

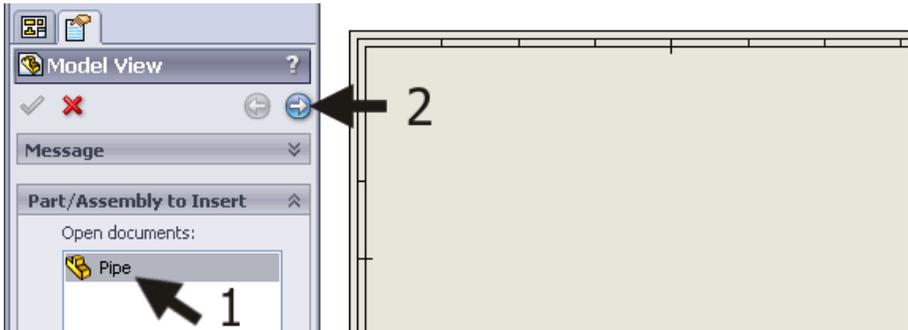
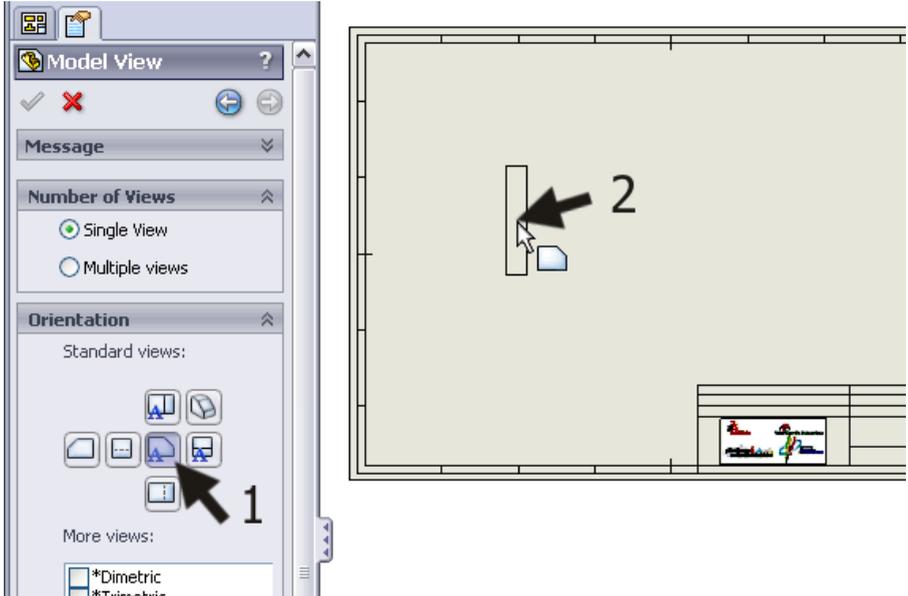
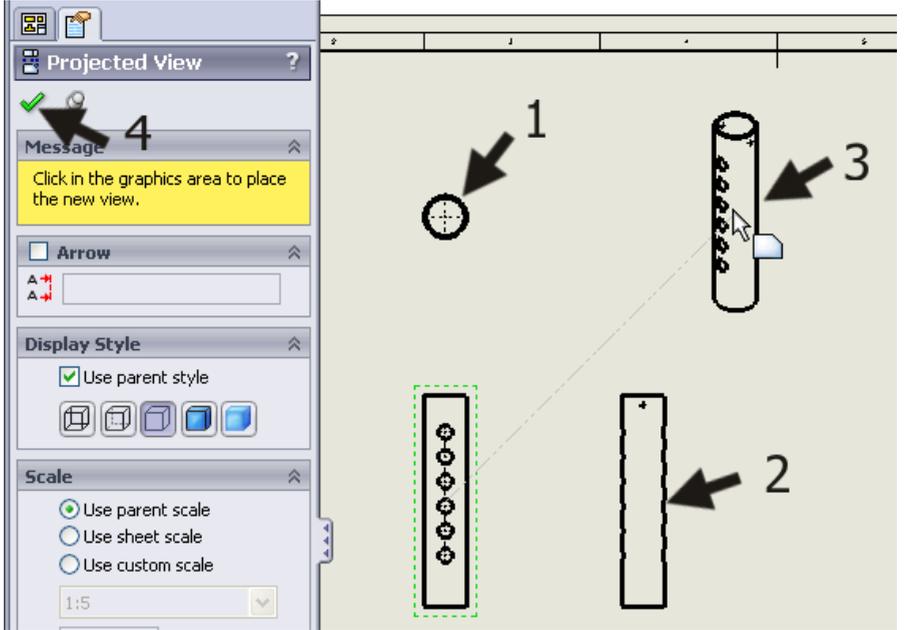


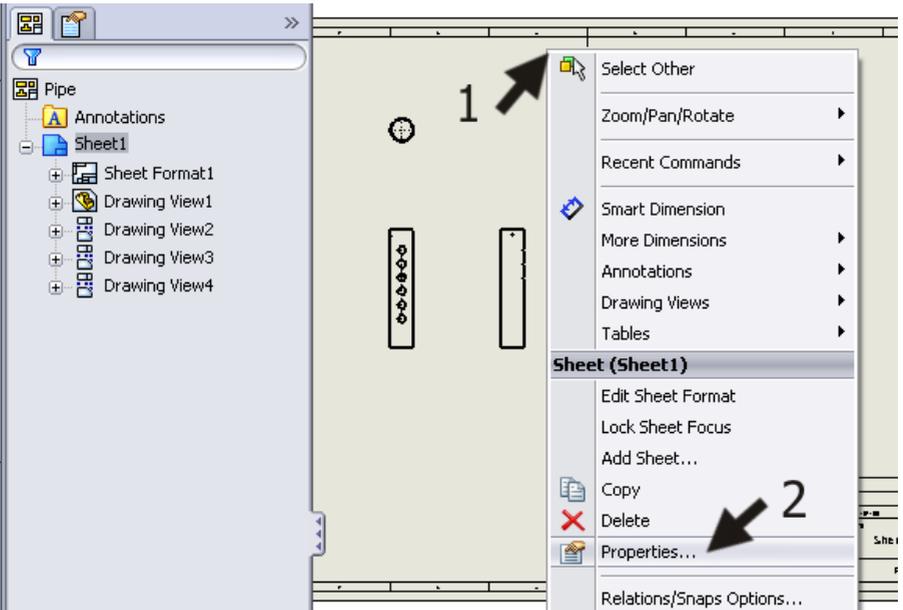
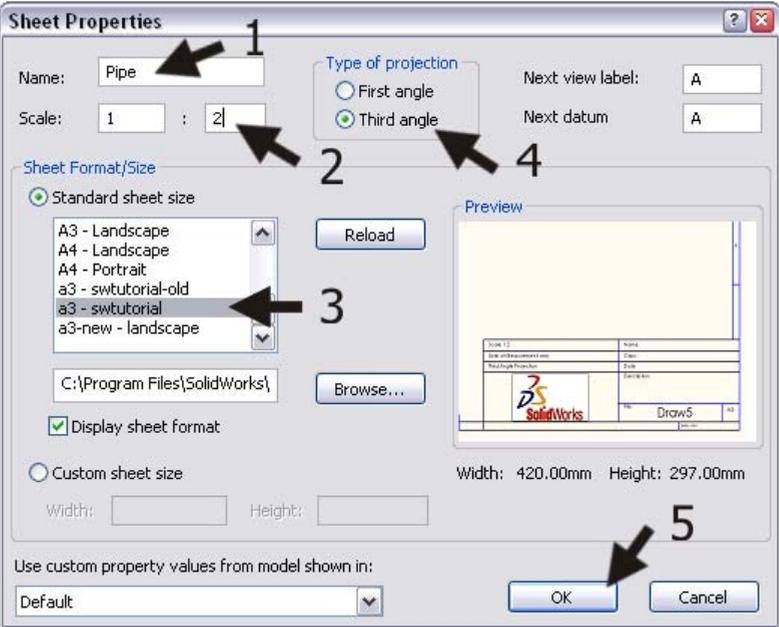
Work plan

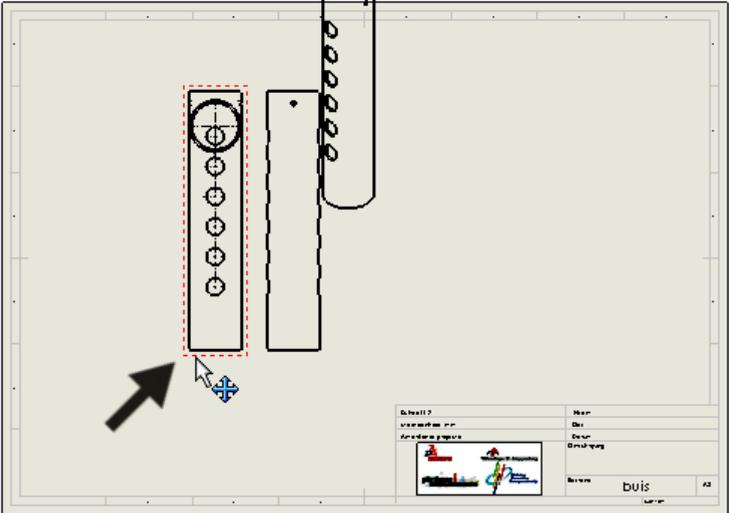
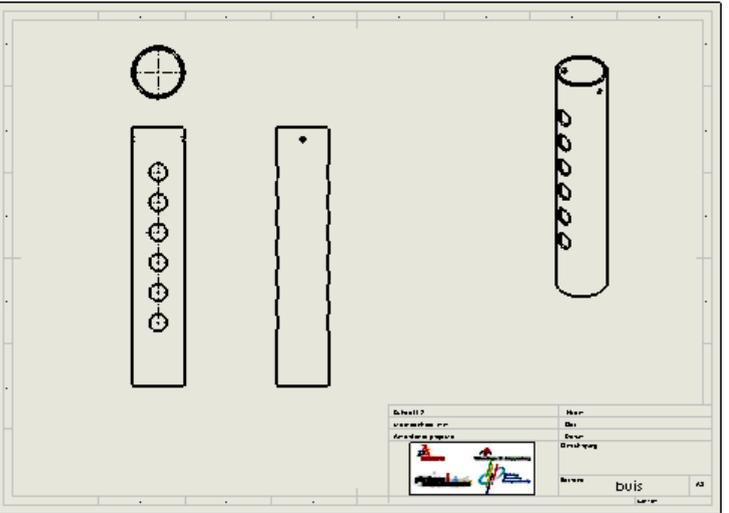
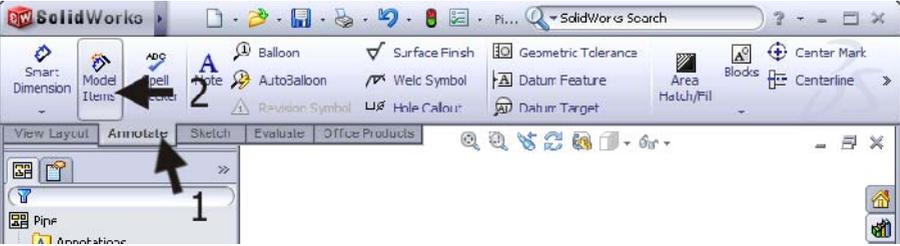
The first part is the inner tube of the axle support. We will put three views on the drawing sheet and an isometric view.

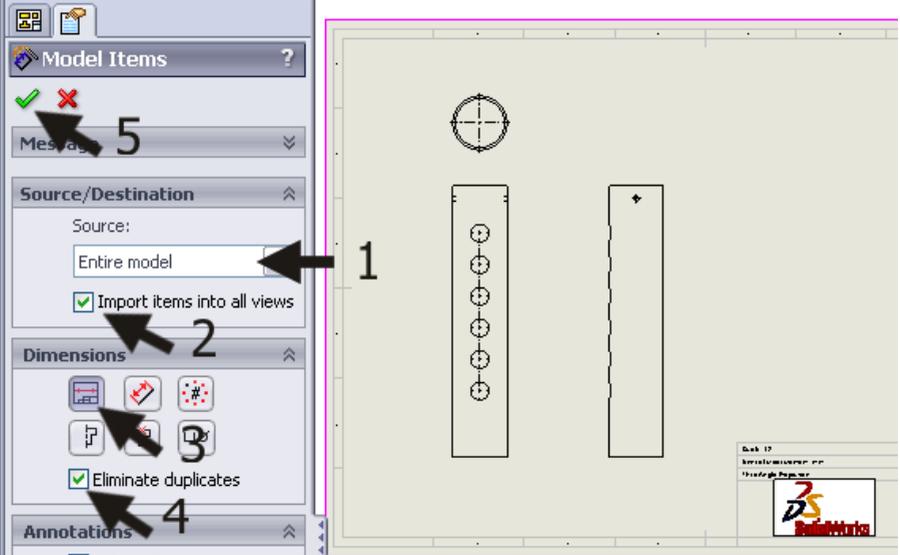


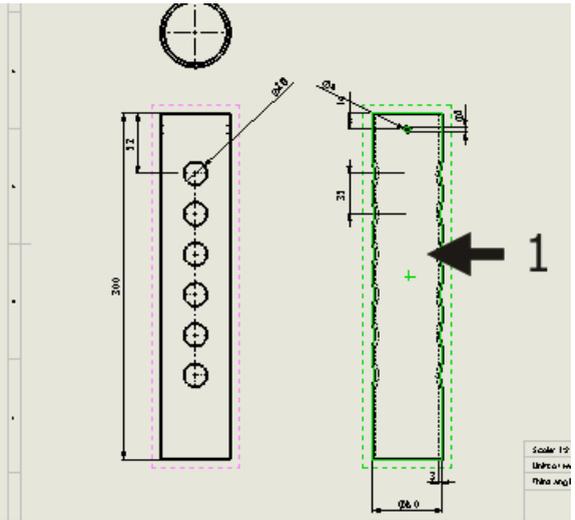
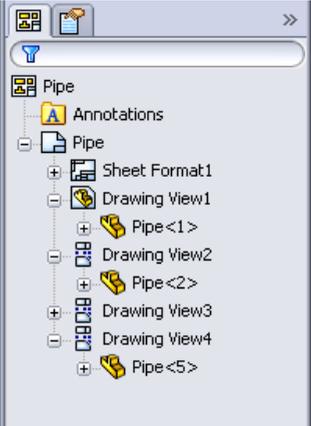
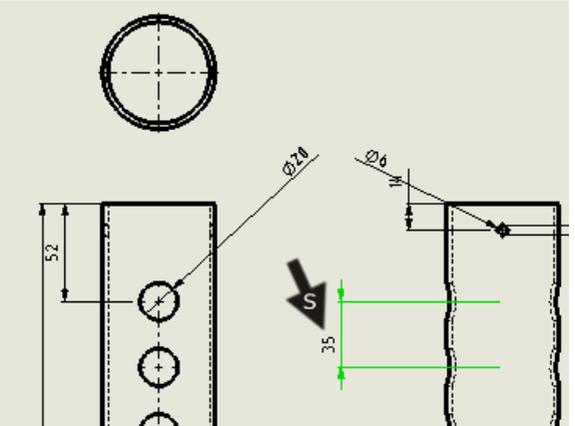
<p>4</p> <p>Select the file 'sw-tutorial' to be your <b>template</b> and click on OK.</p> <p>If this file does not exist, ask your teacher for it. In this file we have made a number of standard settings, so you can start building a proper technical drawing.</p> <p>If you are working at home, you can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

<p>7</p> <p>Set the first view now:</p> <ol style="list-style-type: none"> <li>1. Click on the part 'Pipe'.</li> <li>2. Click on Next.</li> </ol>	
<p>8</p> <p>Make sure the settings in the <b>PropertyManager</b> match the ones on the right.</p> <ol style="list-style-type: none"> <li>1. Select <b>Front</b> for the first view.</li> <li>2. Put it somewhere on the sheet.</li> </ol>	
<p>9</p> <p>After you have positioned the first view, the '<b>Projected View</b>' command will start automatically.</p> <p>Move your cursor around the front view that you put in first. Click three times to set the three views as shown.</p> <p>Click on OK.</p> <p>If the '<b>Projected View</b>' command does not auto-start, click on '<b>Drawings</b>' in the <b>CommandManager</b> and after that on '<b>Projected View</b>'.</p>	
<p>Tip!</p>	<p>There is another method for placing views in a drawing. You can use the <b>Task Pane</b> command. You have done this before in Tutorial 6 (Step 41). As always in SolidWorks: use the method that you prefer!</p>

<p>10</p> <p>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...'.  <span style="color: red;">'Properties...'</span>.</p>	
<p>11</p> <p>Make sure to check the following settings:</p> <ol style="list-style-type: none"> <li>1. 'Name' the drawing: 'Pipe'.</li> <li>2. Select a 'Scale' of '1:2'.</li> <li>3. Paper size is 'a3-sw tutorial'. When this file is not available, ask your teacher for it.</li> <li>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'.</li> <li>5. Click on OK.</li> </ol>	

<p>12</p>	<p>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.</p>	
<p>13</p>	<p>Position the views as shown on the right.</p>	
<p>14</p>	<p>Click on 'Annotate' in the CommandManager, and then on 'Model Items'.</p>	

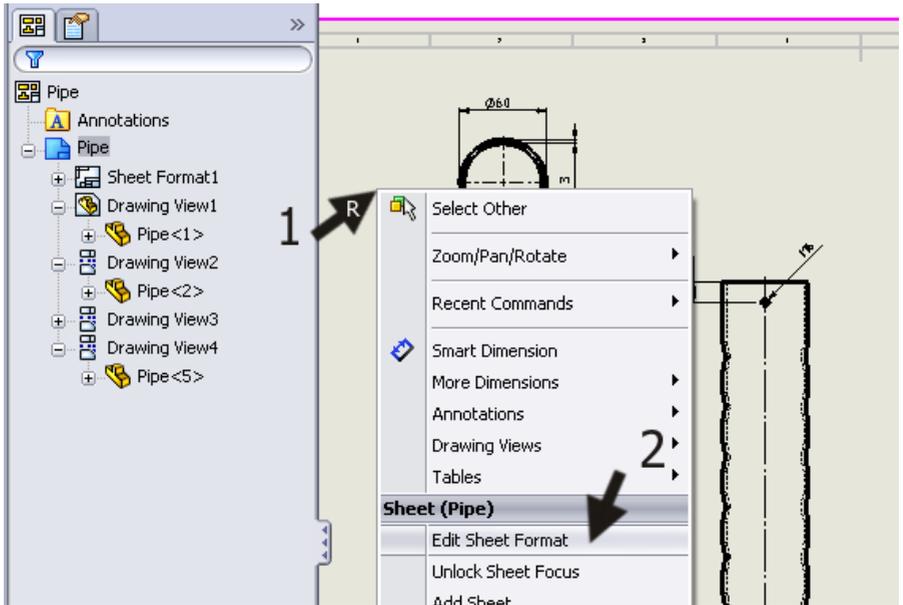
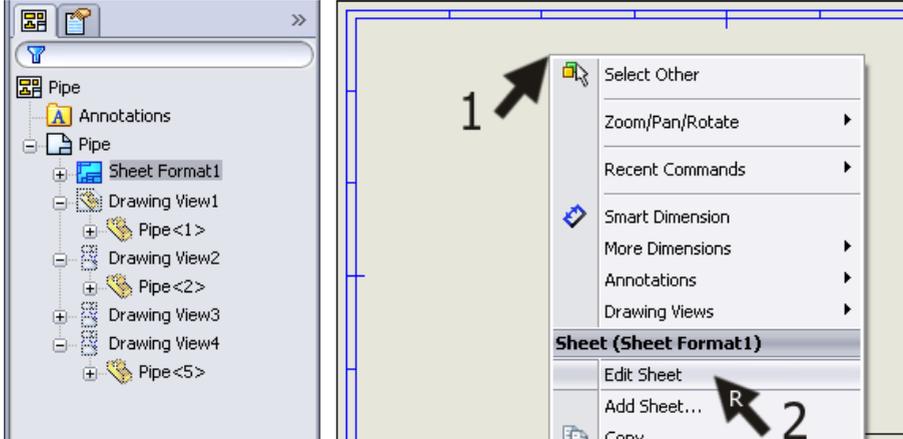
<p>15</p> <p>Set the following features in the <b>PropertyManager</b>:</p> <ol style="list-style-type: none"> <li>1. Select the option 'Entire model' at 'Source/Destination'.</li> <li>2. Check the option 'Import items into all views'.</li> <li>3. Select the first option: 'Marked for drawing' under 'Dimensions'.</li> <li>4. Check the option 'Eliminate duplicates'.</li> <li>5. Click on OK.</li> </ol>	<p>15</p> <p>Set the following features in the <b>PropertyManager</b>:</p> <ol style="list-style-type: none"> <li>1. Select the option 'Entire model' at 'Source/Destination'.</li> <li>2. Check the option 'Import items into all views'.</li> <li>3. Select the first option: 'Marked for drawing' under 'Dimensions'.</li> <li>4. Check the option 'Eliminate duplicates'.</li> <li>5. Click on OK.</li> </ol>	
<p>16</p> <p>The dimensions will now be displayed in the drawing.</p>	<p>16</p> <p>The dimensions will now be displayed in the drawing.</p>	

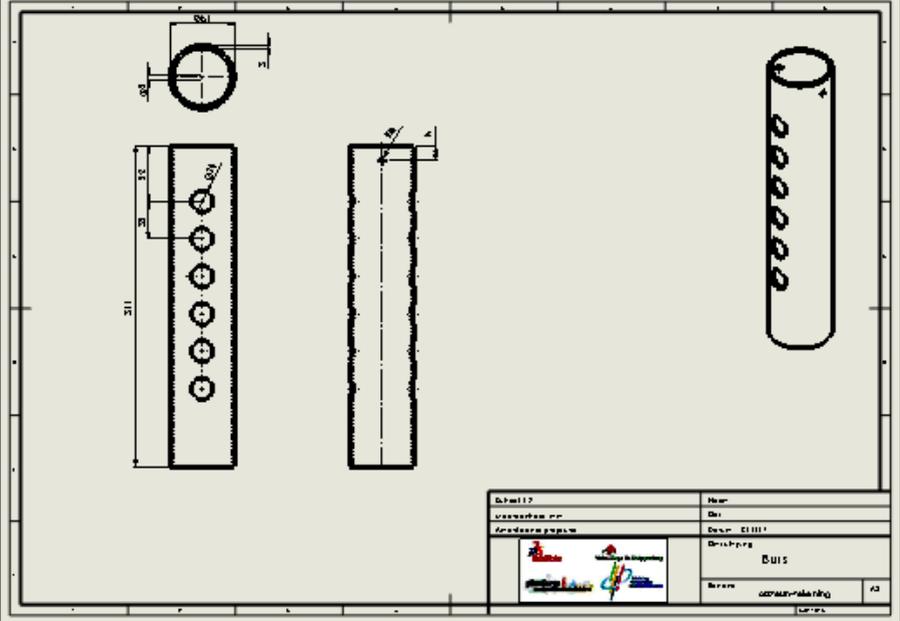
<p>17</p> <p>We will show the invisible lines (dotted lines) in the drawing now.</p> <ol style="list-style-type: none"> <li>1. Click on the side view.</li> <li>2. Select the second option <b>Hidden lines visible</b> under 'Display Style' in the <b>Property-Manager</b>.</li> <li>3. Click on OK.</li> </ol>		
<p>18</p> <p>Do the same for the front view.</p>		
<p>19</p> <p>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.</p> <ol style="list-style-type: none"> <li>1. Drag the size from the right-side view holding the &lt;Shift&gt; key.</li> <li>2. Release the size somewhere in the front view.</li> <li>3. Then, release the &lt;Shift&gt; key on your keyboard.</li> </ol>		

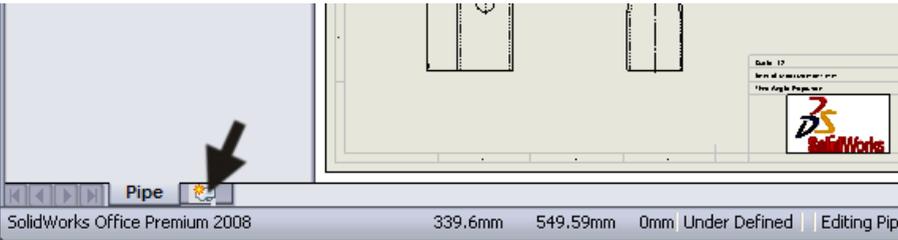
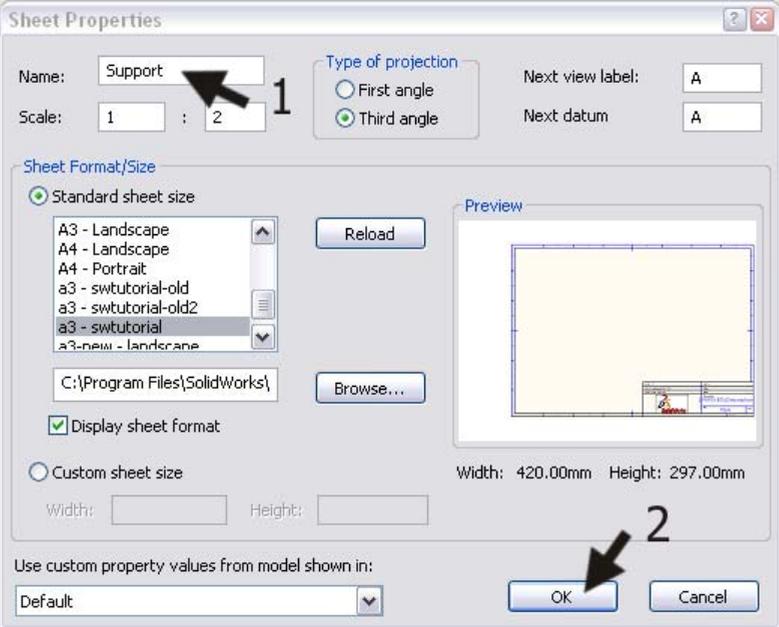
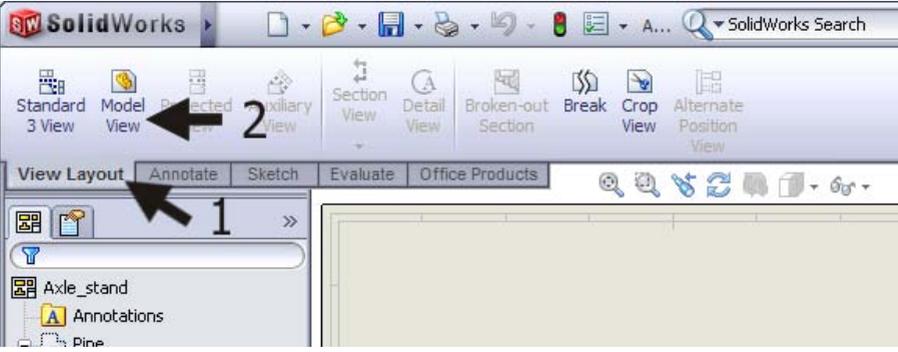


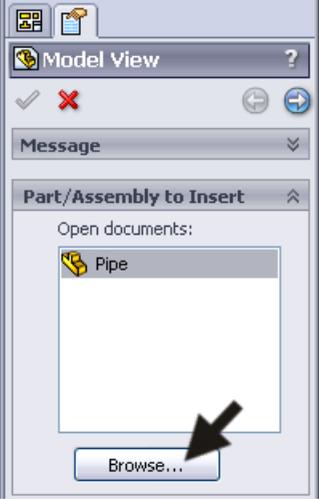
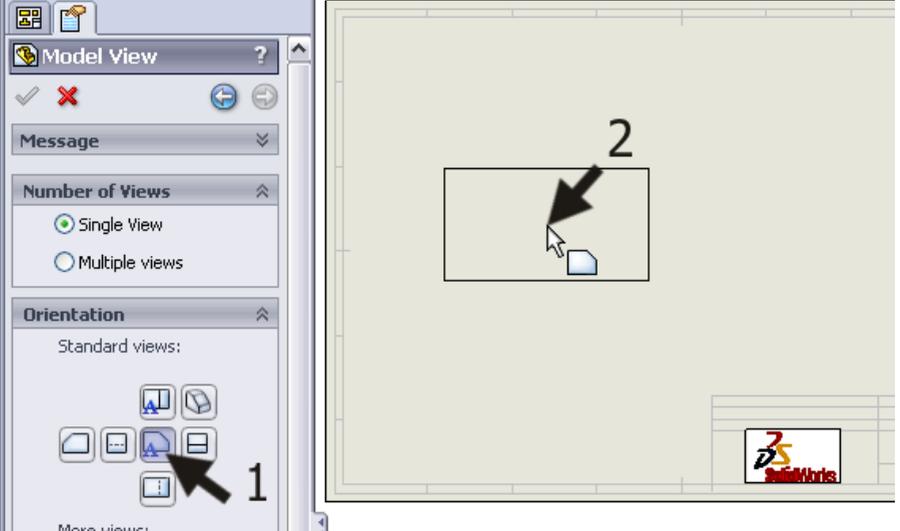
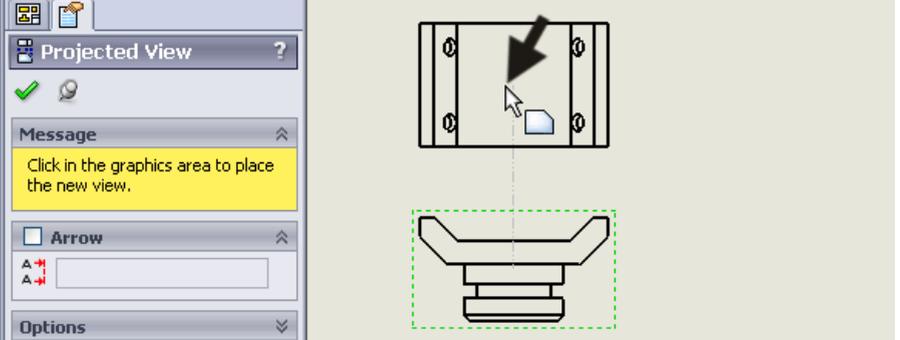
<p>22</p>	<p>Move a few more dimensions like you did in Step 19:</p> <p>The tube diameter (<math>\varnothing 6</math>) and the thickness of the material (3) are moved to the top view.</p> <p>The drawing must look like the illustration on the right.</p>	
<p>23</p>	<p>The size <math>\varnothing 5</math> can be removed.</p> <ol style="list-style-type: none"> <li>1. Click on the size.</li> <li>2. Push the &lt;Del&gt; delete key on the keyboard.</li> </ol>	

<p>24</p> <p>We want to change the size <math>\text{\O}6</math> to M6, because it is a screw thread.</p> <ol style="list-style-type: none"> <li>1. Select the size in the drawing.</li> <li>2. Replace the text '<math>\text{\&lt;MOD-DIAM&gt;}</math>' with the capital 'M' under 'Dimension Text' in the PropertyManager. The field text will read: 'M<math>\text{\&lt;DIM&gt;}</math>'.</li> <li>3. Click on OK.</li> </ol>	
<p>25</p> <p>Finally, we add a centerline to the right-side view.</p> <ol style="list-style-type: none"> <li>1. Select the tube with a click.</li> <li>2. Click on 'Centerline'.</li> </ol> <p>Try to click on another view as well and push the &lt;Esc&gt;key to end the Centerline command.</p>	

<p>26</p> <p>The drawing is now finished. You have to fill in your name in the right bottom corner, in the title block.</p> <ol style="list-style-type: none"> <li>1. Right-click at a random position on the sheet (not on a view or a dimension).</li> <li>2. Click on 'Edit Sheet Format' in the menu.</li> </ol> <p>The drawing will disappear temporarily.</p>	
<p>27</p> <p>Zoom in on the right bottom corner.</p> <p>Double-click on the text field 'Name' and fill in your own name.</p> <p>Do the same with class.</p> <p>The other text fields – such as Date, Description and File – will be filled in automatically by SolidWorks.</p>	
<p>28</p> <p>Right-click somewhere on the page and select 'Edit Sheet' again.</p>	

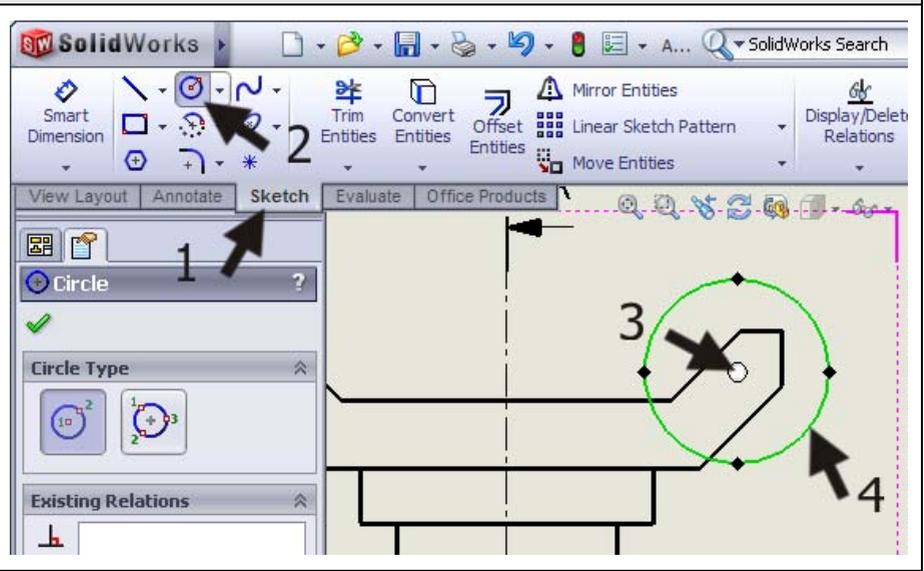
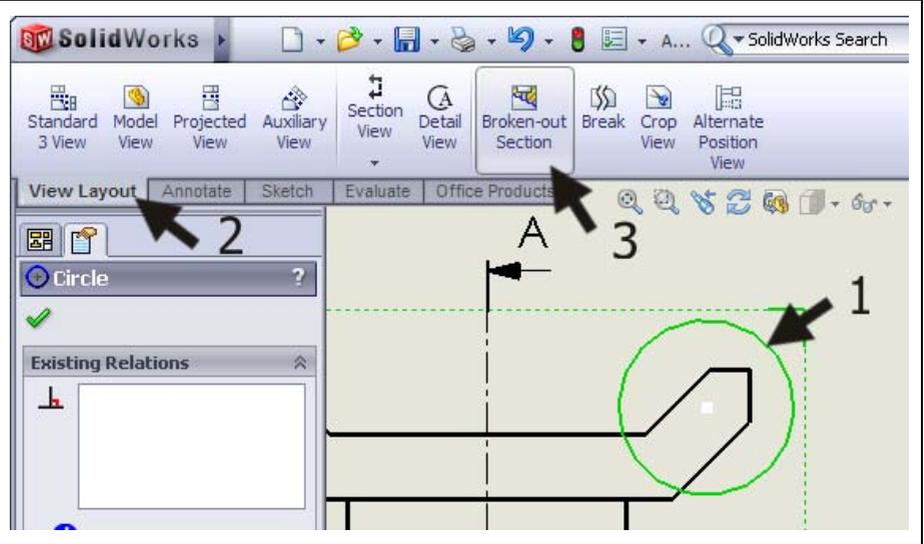
29	Save the file as: Axle_stand.SLDDRW.	
30	<p>Print the drawing.</p> <p>You can find the most important settings for the printer commands in Tutorial 6.</p> <p>Ask your teacher for the right settings for the printer.</p>	
	Work plan	<p>Next we will make a drawing of the support block. In this drawing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

<p>31</p> <p>Add a new sheet to the file first:</p> <ol style="list-style-type: none"> <li>1. Right-click on the tab at the bottom of the screen.</li> <li>2. Select <b>'Add Sheet'</b> in the pop-up menu.</li> </ol> <p>You have two tab sheets now; you can toggle between the drawings if you want to.</p>	
<p>32</p> <p>Right-click somewhere on the new drawing sheet and select <b>'Properties...'</b>.</p> <p>Name the sheet: <b>'Support'</b>.</p> <p>Make sure the settings match those of the first sheet (Step 10).</p>	
<p>33</p> <p>Click on <b>'View Layout'</b> in the <b>CommandManager</b>, and then on <b>'Model View'</b>.</p>	

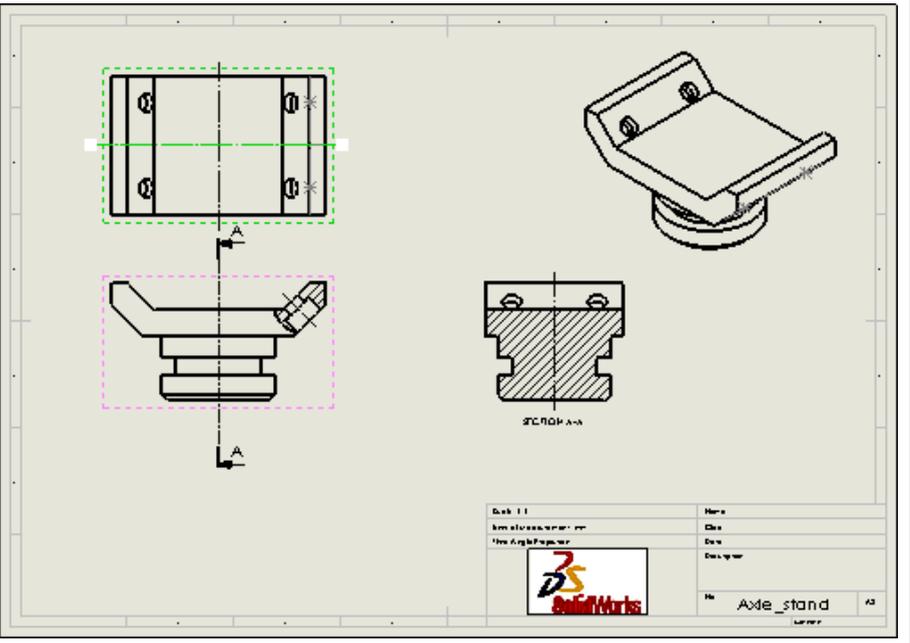
<p>34</p>	<p>When the part 'Support' is opened, select it in the list in the <b>PropertyManager</b>. If not, click on 'Browse...' and find the file on your hard disk or on your memory stick.</p>	
<p>35</p>	<p>Click on the sheet to place the front view.</p>	
<p>36</p>	<p>The command 'Projected View' will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

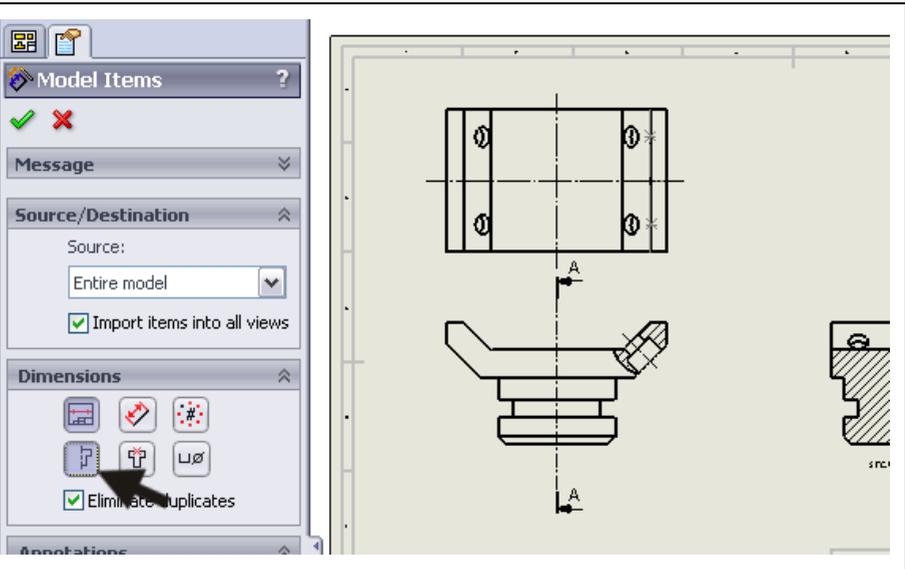
<p>37</p> <p>Let's make a cross-cut!</p> <ol style="list-style-type: none"> <li>1. Click on 'Section View' in the CommandManager.</li> </ol> <p>Draw a cross-cut line. Be careful. Do this very accurately!</p> <ol style="list-style-type: none"> <li>2. Set the cursor on top of the midpoint of the upper horizontal line, but do NOT click!</li> </ol>		
<p>38</p> <p>Move the cursor upward; you can see a blue dotted line.</p> <p>Click above the view.</p>		
<p>39</p> <p>Move the cursor downward now and click underneath the view.</p>		
<p>40</p> <p>Next, the cross-cut will appear and you can place it beside the view.</p>		
<p>Tip!</p>		<p>When you accidentally shift the cross-cut line, colored shading will appear</p>

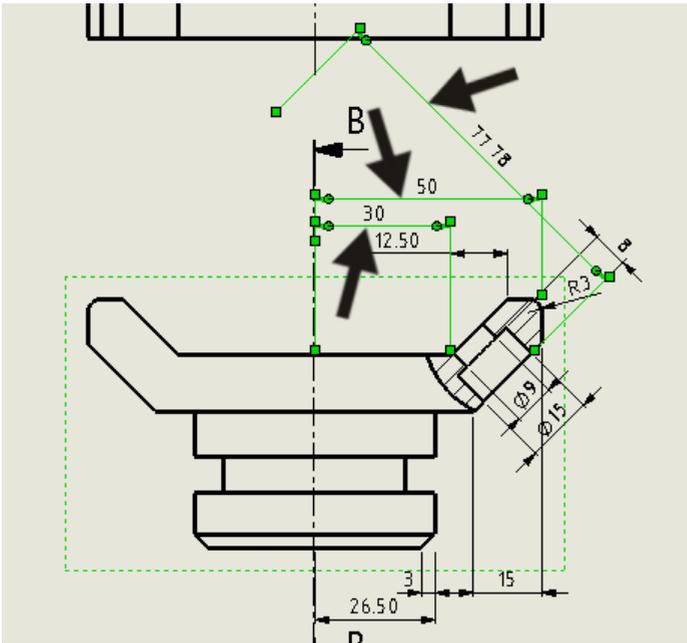
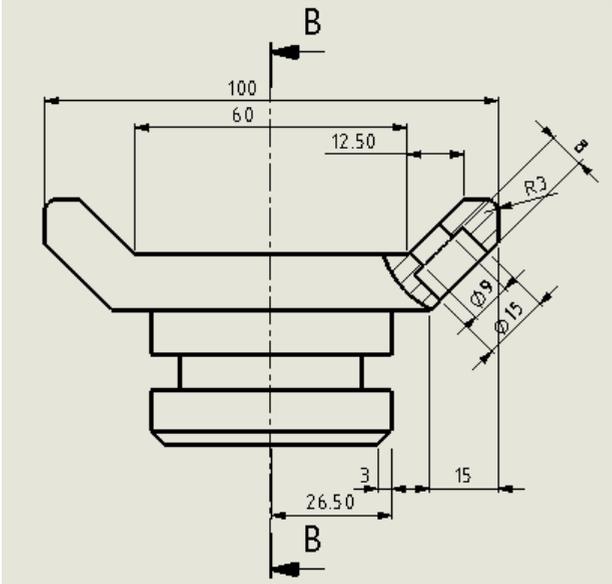
in the cross-cut drawing. This indicates that the model has to be updated.  
 In such a case, click on the **Rebuild** button in the standard toolbar. The colored hatching will disappear.

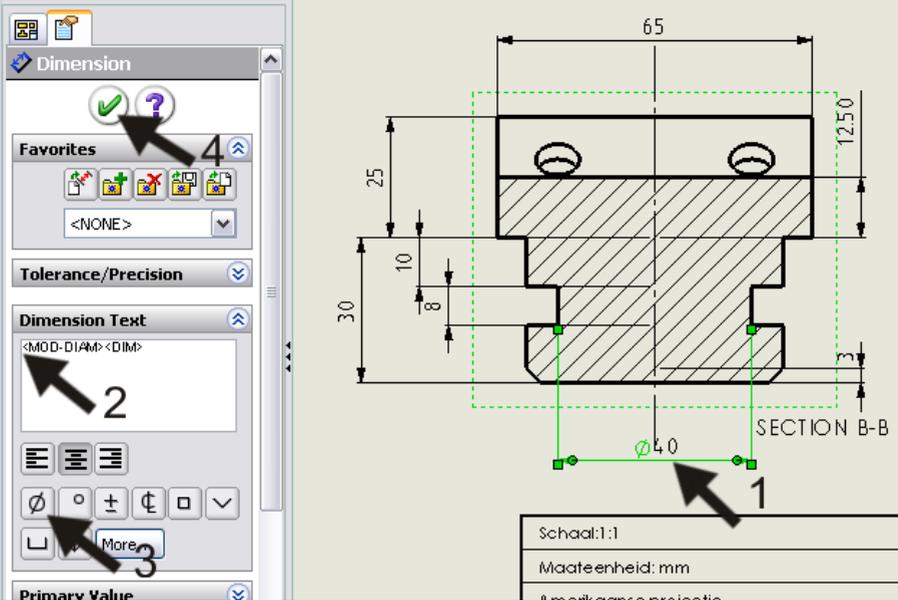
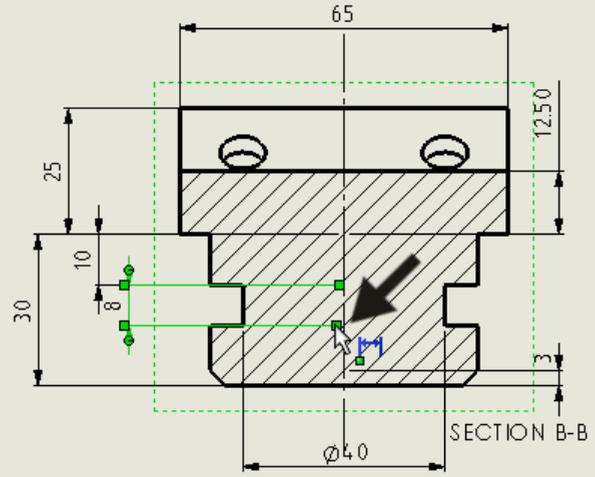
<p>41</p> <p>To get a better view of the countersink hole, we will open a part of the front view.</p> <p>Click on <b>'Sketch'</b> and after that on <b>Circle</b> in the <b>CommandManager</b>.</p> <p>Set the circle just about the same as in the illustration on the right.</p>	
<p>42</p> <ol style="list-style-type: none"> <li>1. Make sure the circle is selected (it turns green).</li> <li>2. Click on <b>'View Layout'</b> in the <b>CommandManager</b>.</li> <li>3. Click on <b>'Broken-out Section'</b>.</li> </ol>	

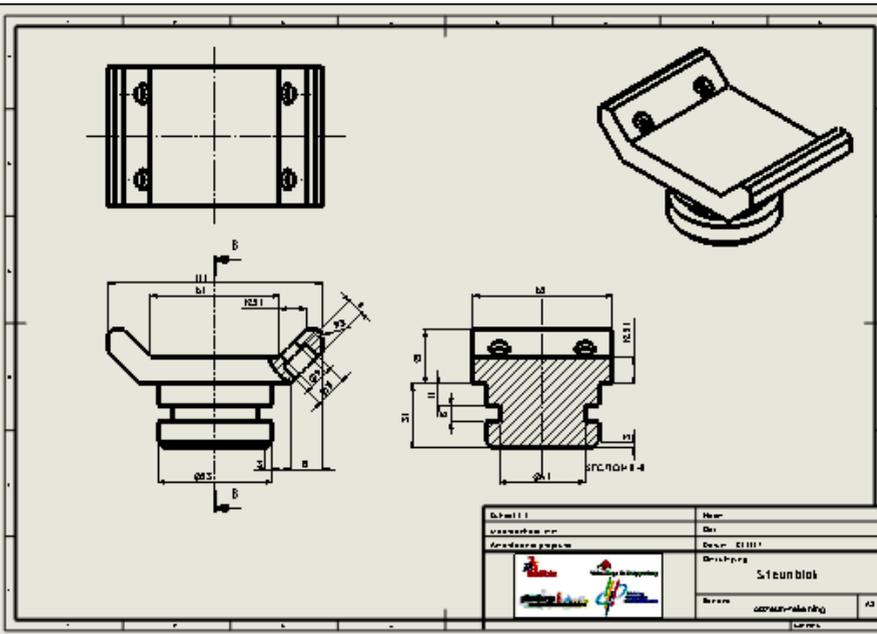
<p>43</p>	<ol style="list-style-type: none"> <li>1. Check the option 'Preview' in the Property-Manager.</li> <li>2. Click on the edge of the hole in the top view. The cross-cut will run through here.</li> <li>3. If the preview looks good, click on OK.</li> </ol>	
<p>44</p>	<p>To put a centerline in the hole use the following steps:</p> <ol style="list-style-type: none"> <li>1. Select the first edge from the hole.</li> <li>2. Hold the &lt;Ctrl&gt; key and select the second edge from the hole.</li> <li>3. Click on the tab 'Annotate' in the Command-Manager.</li> <li>4. Click on 'Centerline'.</li> </ol> <p>The centerline is a bit short now, but you can drag the end to extend it.</p>	

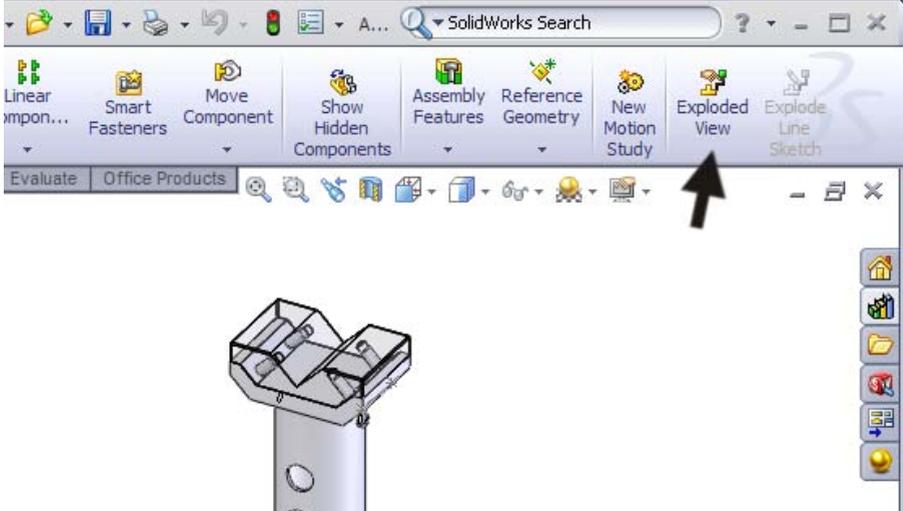
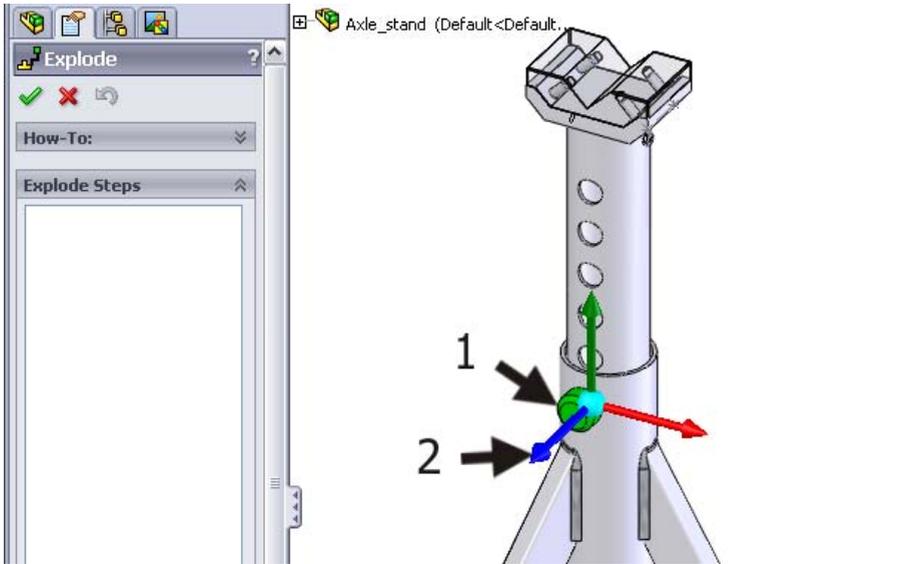
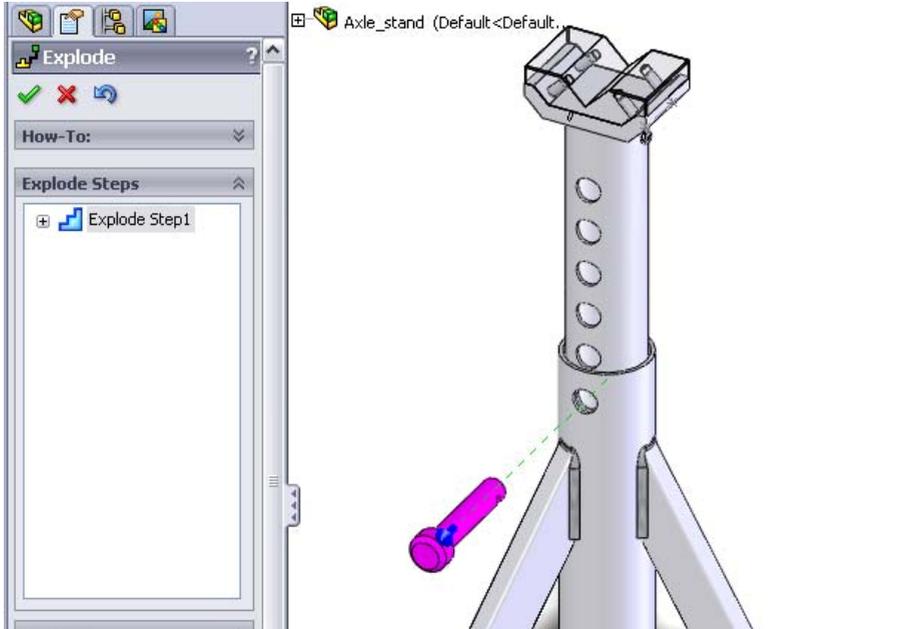
<p>45</p> <p>Add the other <b>centerlines</b> too, so the drawing will end up looking like the illustration on the right.</p>	
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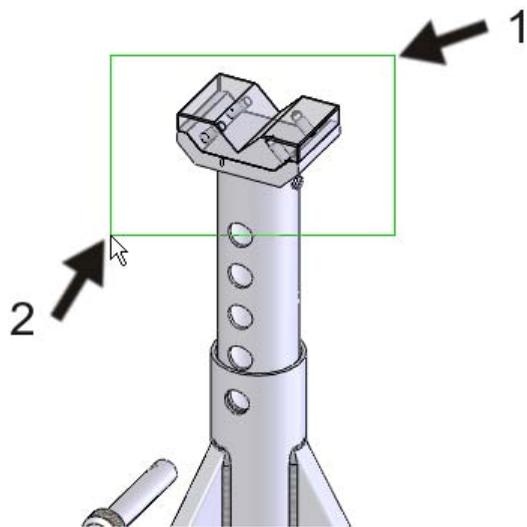
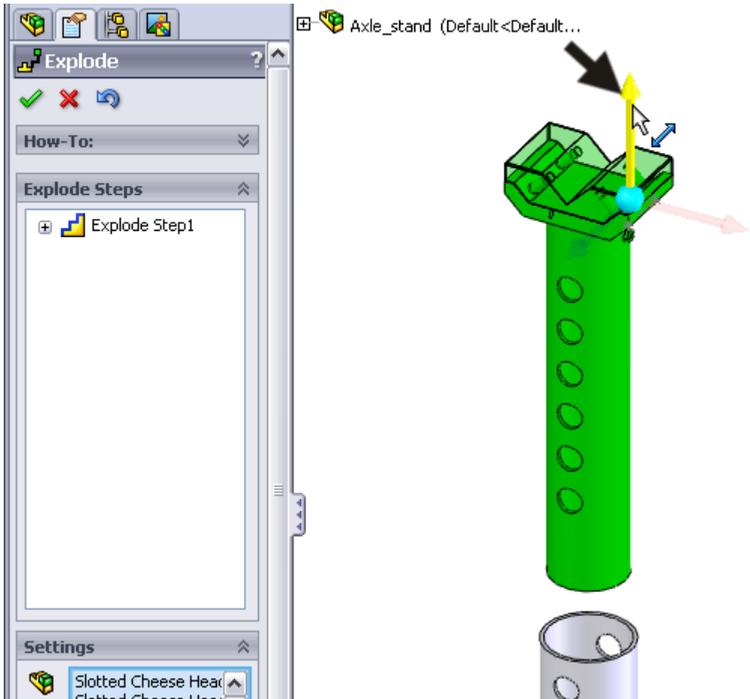
<p>46</p> <p>Set the dimensions in the drawing now.</p> <p>Click on <b>'Annotate'</b> in the <b>CommandManager</b> and then on <b>'Model Items'</b>.</p> <p>Use the same setting as in the last drawing (Step 14). Make sure that the option <b>Hole Wizard Profiles</b> is also checked.</p> <p>Click on OK.</p>	
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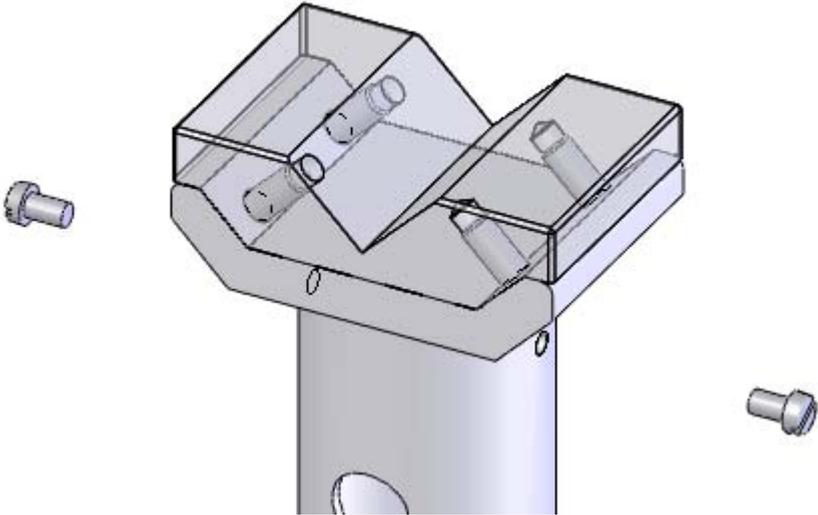
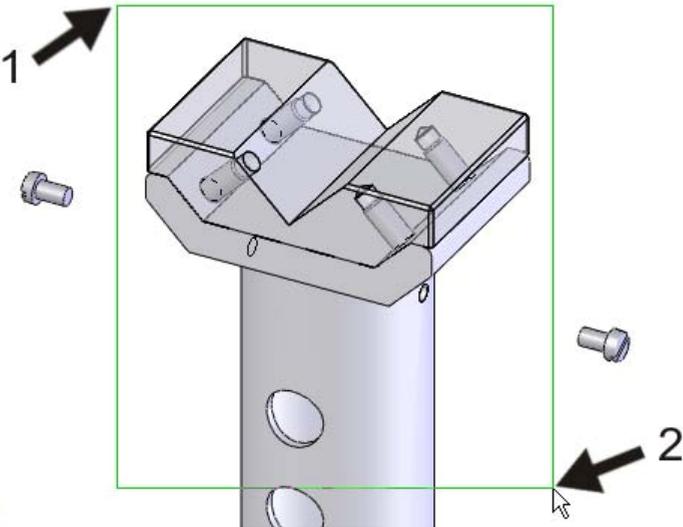
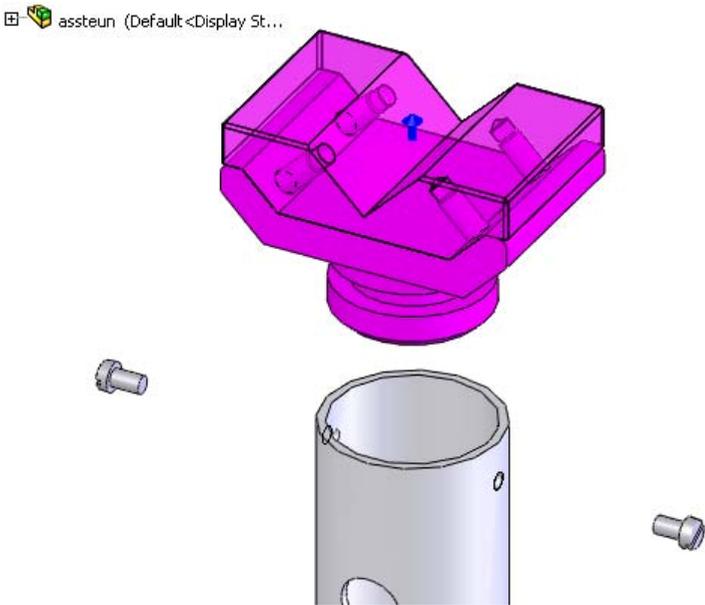
<p>47</p> <p>Shift the dimensions where necessary so your drawing looks orderly.</p> <p>Select the three dimensions as shown on the right.</p> <p>Push the &lt;Del&gt; delete key on the keyboard to remove them.</p>	
<p>48</p> <p>Next, set two new dimensions. Click on 'Smart Dimension' in the Command-Manager and set the dimensions of '60' and '100' like in the illustration.</p> <p>You have also used 'Smart Dimension' in sketches so you should be familiar with this command.</p>	

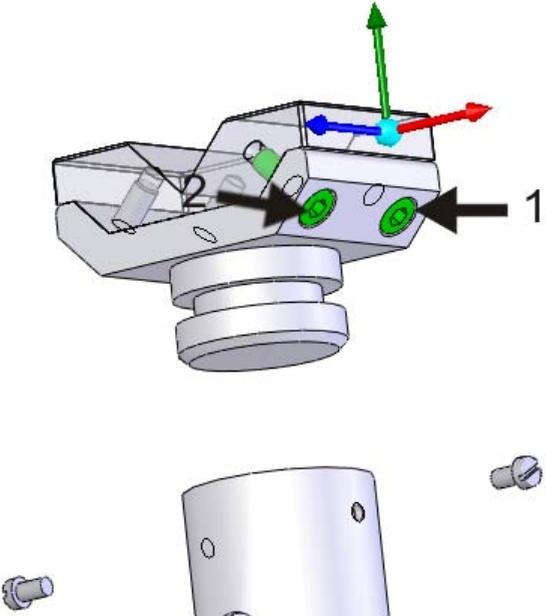
<p>49</p> <p>Replace the size R20 with R40 in the same way.</p> <p>Follow the next few steps to set a <math>\varnothing</math> symbol in front of the actual size:</p> <ol style="list-style-type: none"> <li>1. Select the dimension.</li> <li>2. Set the cursor at 'Dimension Text' in front of the existing text '&lt;DIM&gt;' in the PropertyManager.</li> <li>3. Click on the diameter-symbol. In the text field it reads: '&lt;MOD-DIAM&gt;&lt;DIM&gt;'.</li> <li>4. Click on OK.</li> </ol>	
<p>50</p> <p>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.</p>	
<p>Tip!</p>	<p>Notice that we have inserted dimensions in the drawing in two different ways:</p> <ol style="list-style-type: none"> <li>1. By importing them from the 3D model.</li> <li>2. By putting them in the drawing manually with the <b>Smart Dimension</b> command.</li> </ol> <p>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 <b>Driving Dimensions</b>.</p> <p>It is not possible to change a manually placed dimension. If you double-click on those, nothing happens. These are <b>Driven Dimensions</b>.</p>

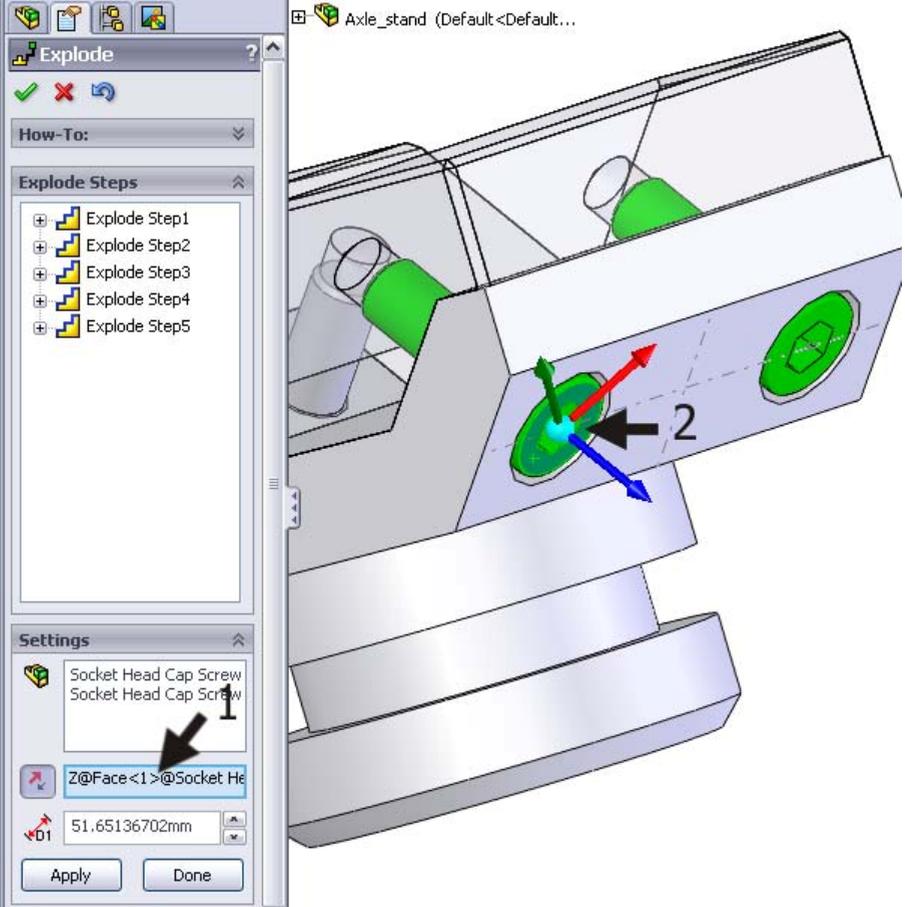
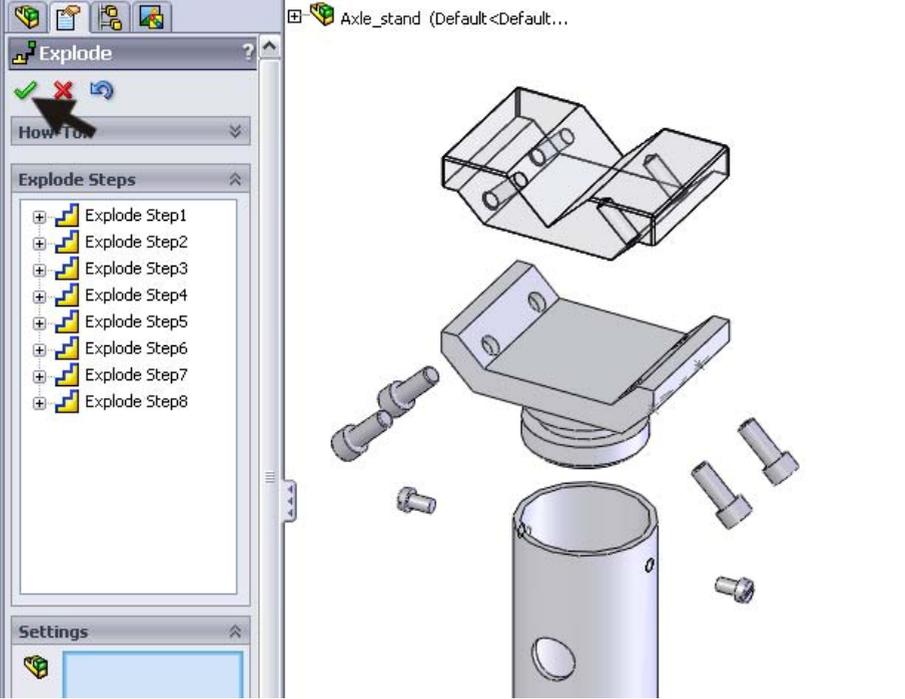
<p>51</p> <p>Fill in the right bottom corner of the drawing as you have done before (in Steps 26 to 28).</p> <p>Click on Save to save the file.</p>		
	<p>Work plan</p>	<p>Finally, we will make the drawing of the <b>assembly</b>. For this we use one of the views and an exploded view. To put an exploded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

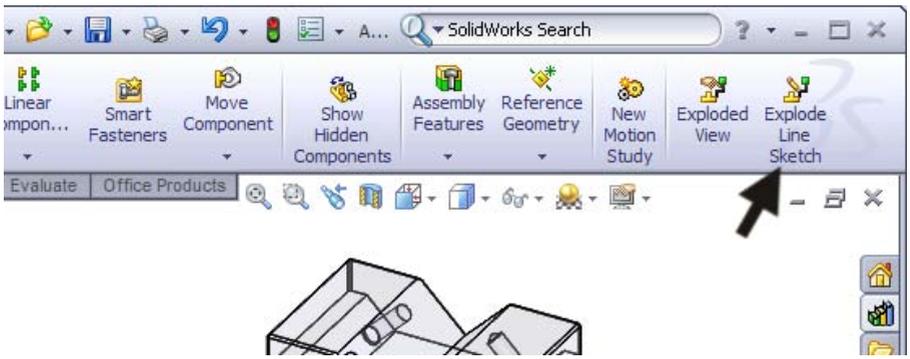
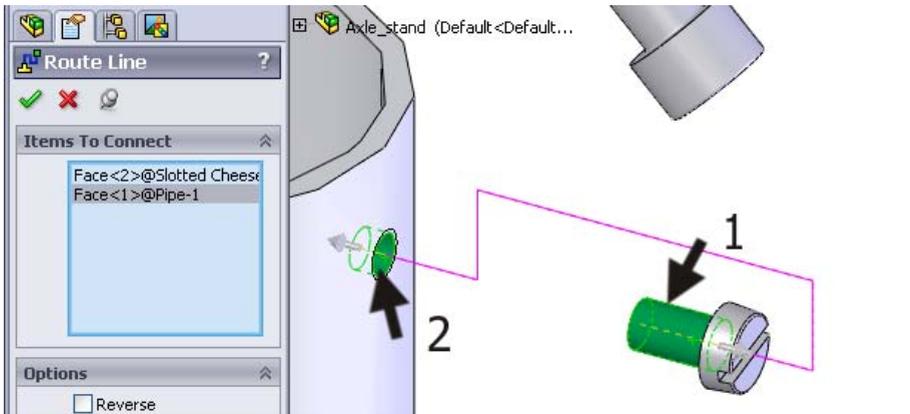
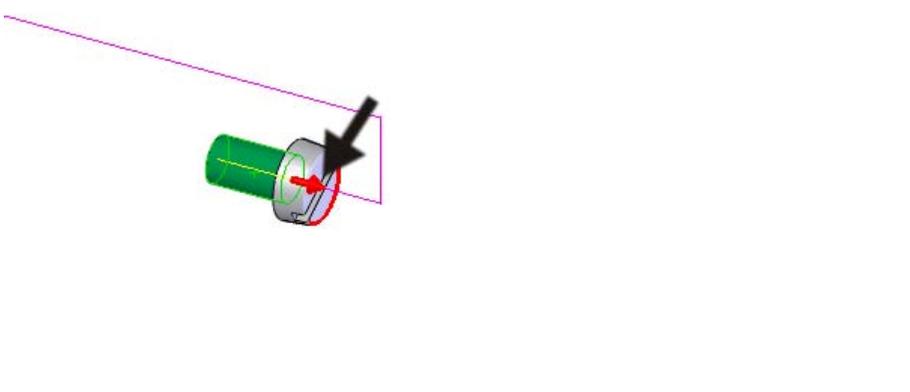
<p>52</p> <p>Open the file Axle_stand.SLDASM.</p> <p>Click on 'Exploded View' in the CommandManager.</p>		 <p>The image shows the SolidWorks CommandManager ribbon. The 'Exploded View' button, which features an icon of a disassembled assembly, is highlighted with a black arrow. Other buttons visible include 'Linear Motion...', 'Smart Fasteners', 'Move Component', 'Show Hidden Components', 'Assembly Features', 'Reference Geometry', 'New Motion Study', and 'Explode Line Sketch'.</p>
<p>53</p> <p>Click on the pin.</p> <p>At the pin three arrows appear and you can now drag this part in three different directions.</p>		 <p>The image shows the 'Explode' dialog box on the left, which is currently empty. On the right, a 3D model of the axle stand is shown with a pin inserted into a hole. Three arrows originate from the pin: a green arrow pointing upwards, a blue arrow pointing to the left, and a red arrow pointing to the right. The numbers '1' and '2' are placed next to the green and blue arrows, respectively, with black arrows pointing to them.</p>
<p>54</p> <p>Drag the part using the blue arrow, so the pin will end up beside the assembly.</p> <p>Click somewhere beside the model to un-select the pin.</p>		 <p>The image shows the same 3D model as in the previous step. The pin has been moved to the side of the axle stand. A dashed green line indicates the original position of the pin. The 'Explode' dialog box on the left now shows 'Explode Step1' with a blue arrow icon next to it.</p>

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

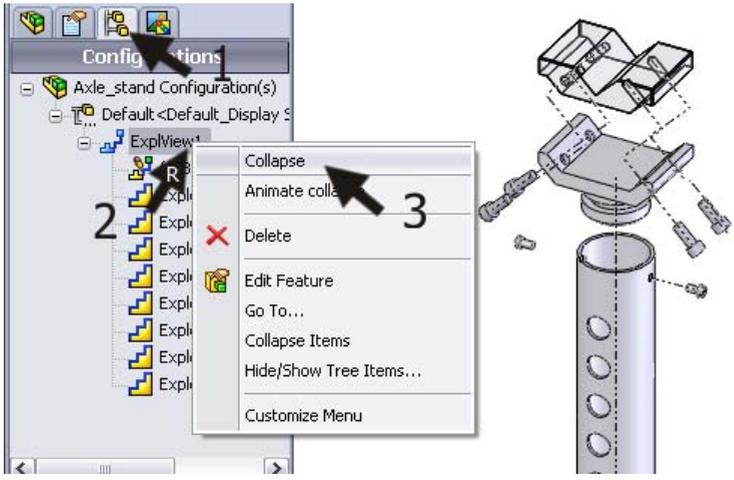
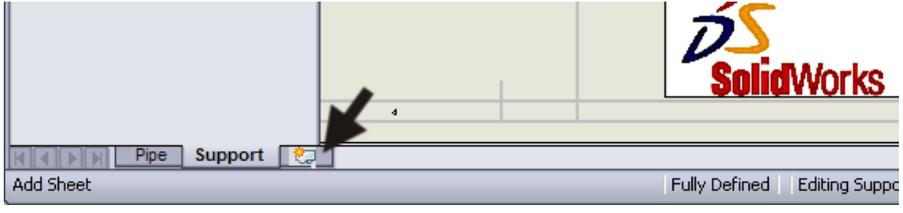
<p>57</p>	<p>Drag the two little screws just below the support block one by one until they are just outside of the model.</p> <p>You can easily rotate the model during this operation, but remember to put it back into the trimetric position. This is the only way to you will get a clear idea about how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

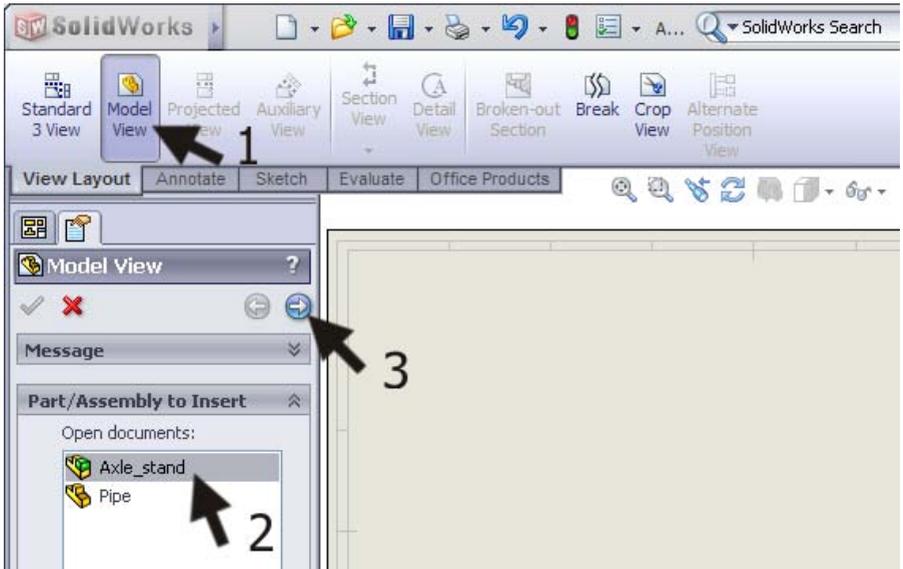
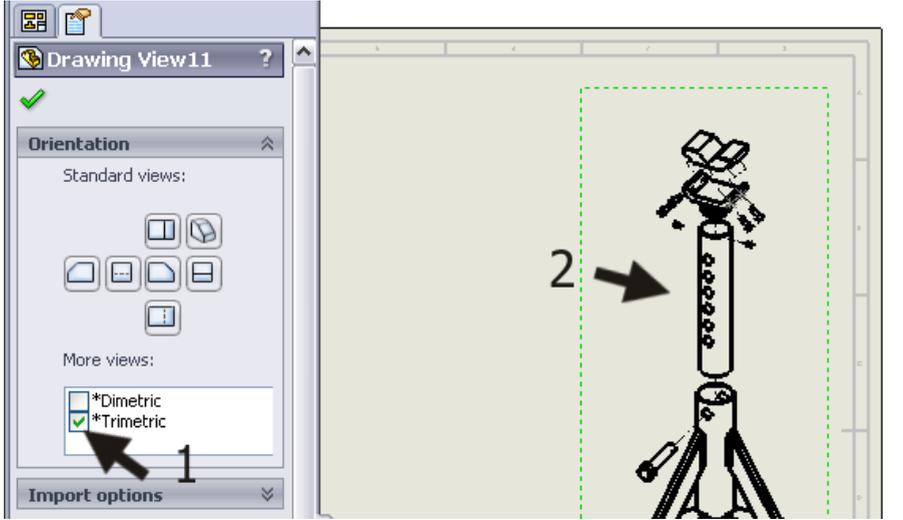
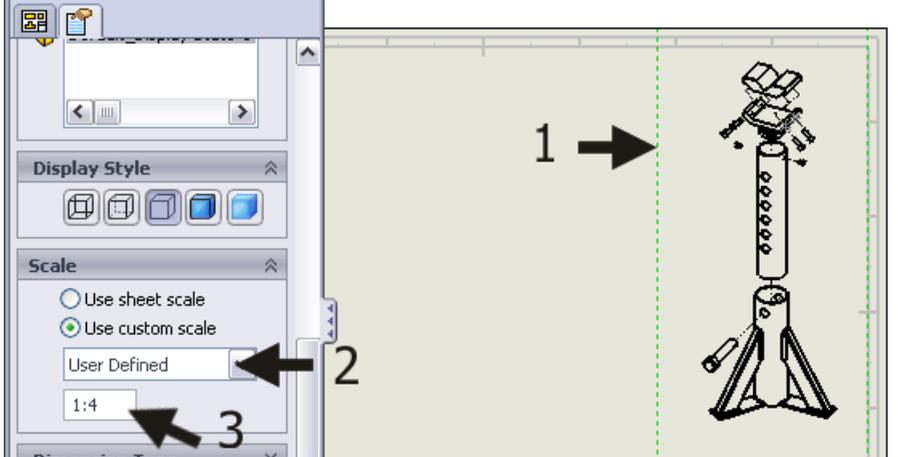
	<p>Tip!</p>	<p>If the part is not directly in the right spot, you can always click on it a second time and drag it to a new position. However, this will create a new 'Explode Step' and make your model more complex.</p> <p>It is much better to find the step you want to change in the <b>PropertyManager</b> and then click on it. A blue arrow will appear on the part, and you can change its position by dragging the blue arrow.</p>
60	<p>Rotate the model a little so you can see two of the screws in the bottom of the support block.</p> <p>Select the two screws.</p>	 <p>The image shows a 3D CAD model of an axle support assembly. The main part is a grey, multi-faceted support block with a central cylindrical hole. Two screws are highlighted in green on the bottom surface of the support block. A black arrow labeled '1' points to the right screw, and another black arrow labeled '2' points to the left screw. Below the main assembly, the exploded parts are shown: a cylindrical axle, a support block, and two screws. A 3D coordinate system with red, green, and blue axes is visible in the upper right of the main assembly view.</p>

<p>61</p> <p>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:</p> <ol style="list-style-type: none"> <li>1. Click on the field <b>Explode Direction</b> in the <b>PropertyManager</b>.</li> <li>2. Click on a plane from the screw.</li> </ol> <p>The arrows now change direction, and you can shift the screws in the right direction.</p>	
<p>62</p> <p>Move the other two screws in the same way.</p> <p>Raise the insert.</p> <p>The parts are all in the right position now.</p> <p>Click on OK in the <b>PropertyManager</b>.</p>	

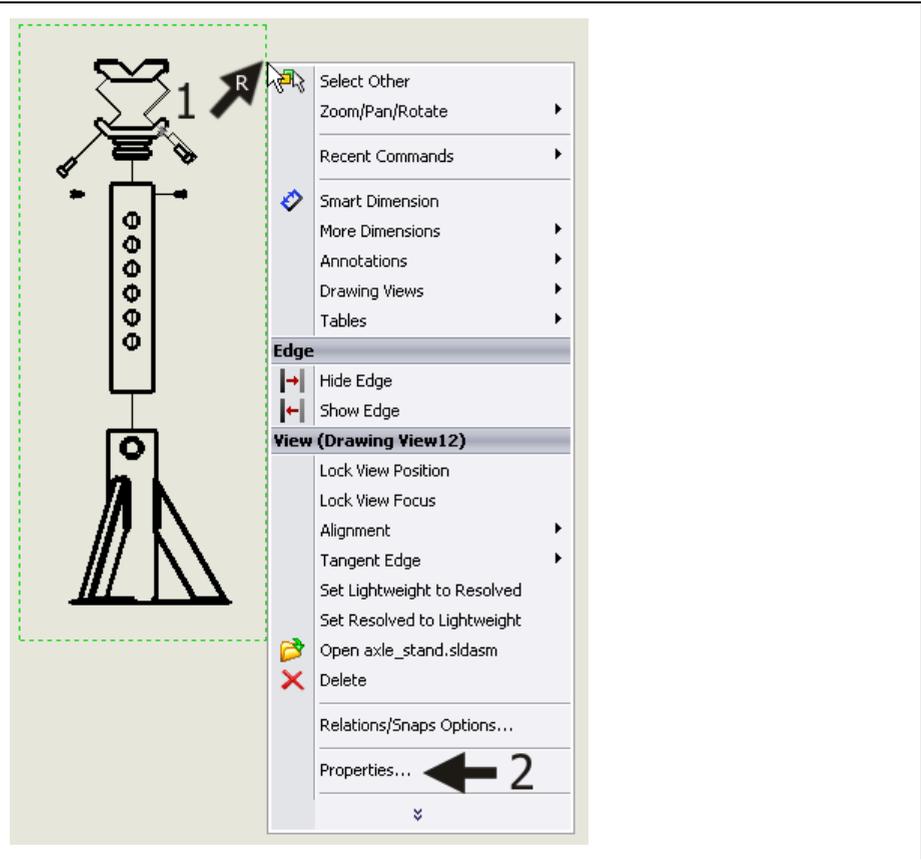
<p>63</p>	<p>To get a better idea about how the parts of the product fit together, the parts are often connected with lines.</p> <p>Click on 'Explode Line Sketch' in the Command-Manager to do so.</p>	
<p>64</p>	<p>Select the two planes as shown on the right.</p>	
<p>65</p>	<p>You can see that the line 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.</p> <p>When the line is ok, click on OK in the <b>PropertyManager</b>.</p>	

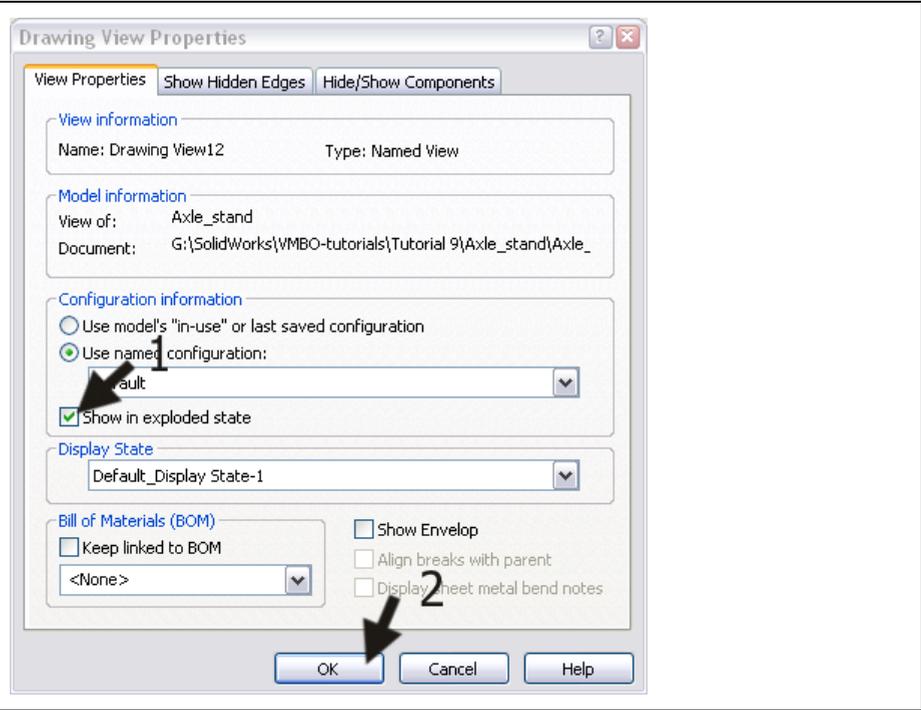
<p>66</p>	<p>Make a similar line for one of the screws in the support block. Select the three planes as shown.</p>	
<p>67</p>	<p>Draw all connection lines in the exploded view in the same way.</p>	

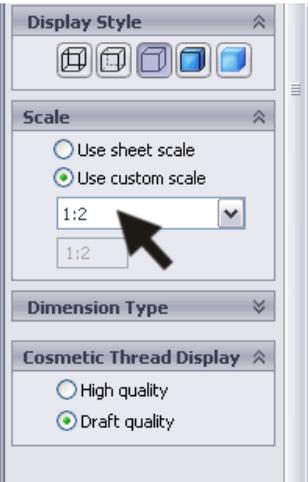
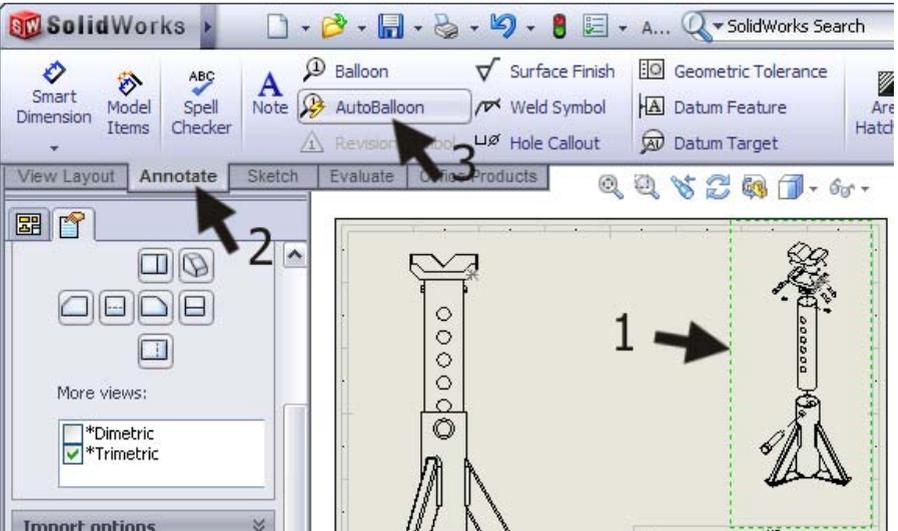
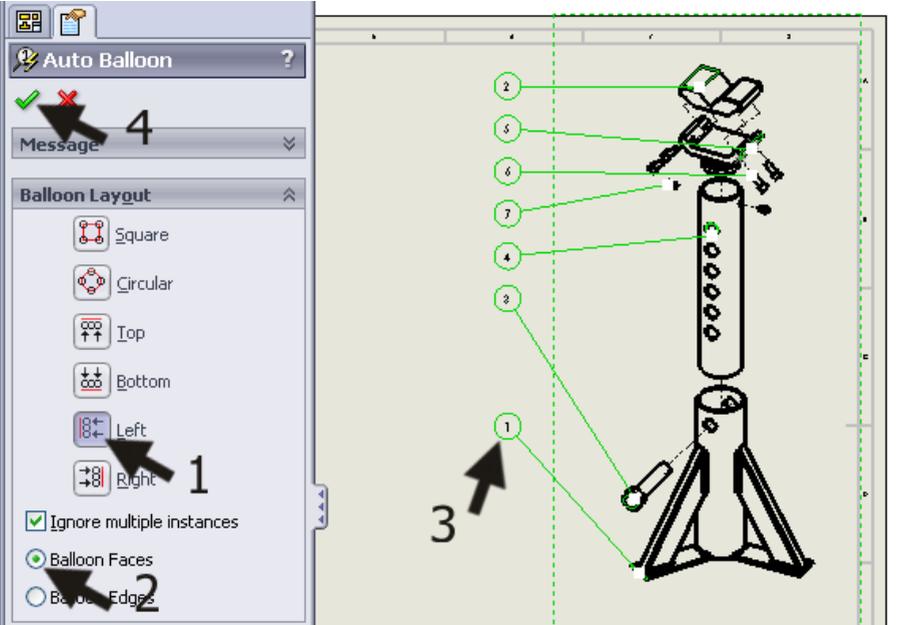
<p>68</p> <p>The <b>assembly</b> has now turned into an <b>Exploded View</b>. But how do we return to the normal <b>assembly</b>?</p> <ol style="list-style-type: none"> <li>1. Go to the <b>ConfigurationManager</b>.</li> <li>2. Right-click on <b>'ExplView1'</b>.</li> <li>3. Select <b>'Collapse'</b>.</li> </ol> <p>To return to the exploded view again, select <b>'Explode'</b> in the same menu.</p> <p>Try the option <b>'Animate collapse/explode'</b>. You will see the parts moving away from and toward each other.</p>	
<p>69</p> <p>Make sure the <b>assembly</b> is exploded and save the file.</p> <p>Return to the drawing again that you were working in.</p> <p>Push the capital 'R' on the keyboard.</p> <p>Click on <b>Axle_stand.SLDDRW</b>.</p>	
<p>70</p> <p>Add a drawing sheet to the file.</p> <p>Click on <b>Add Sheet</b>.</p>	

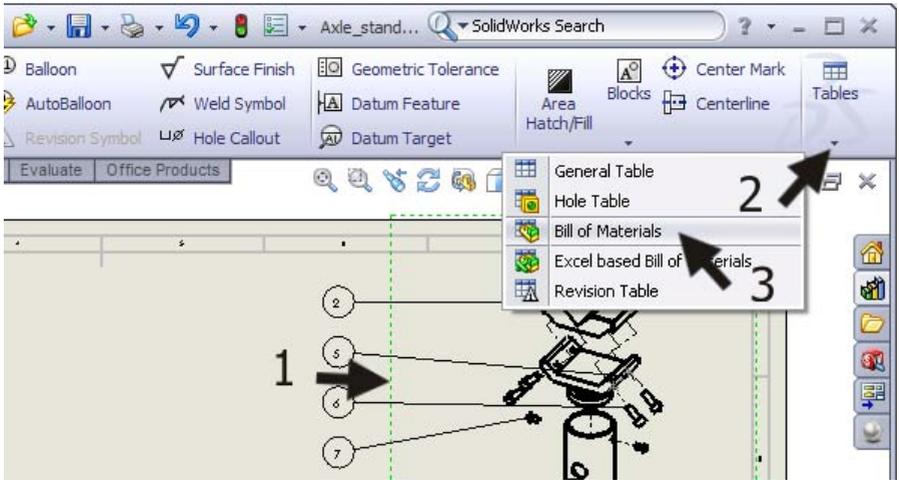
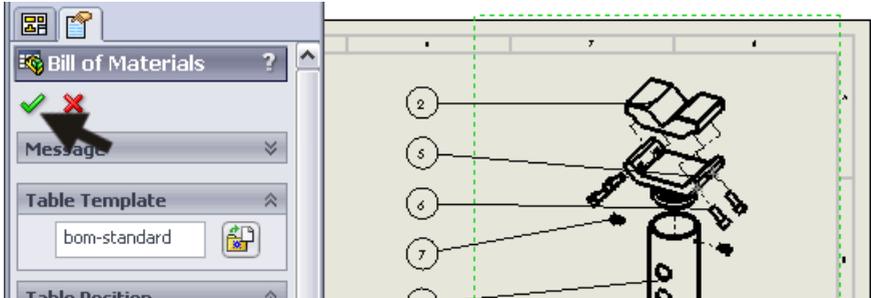
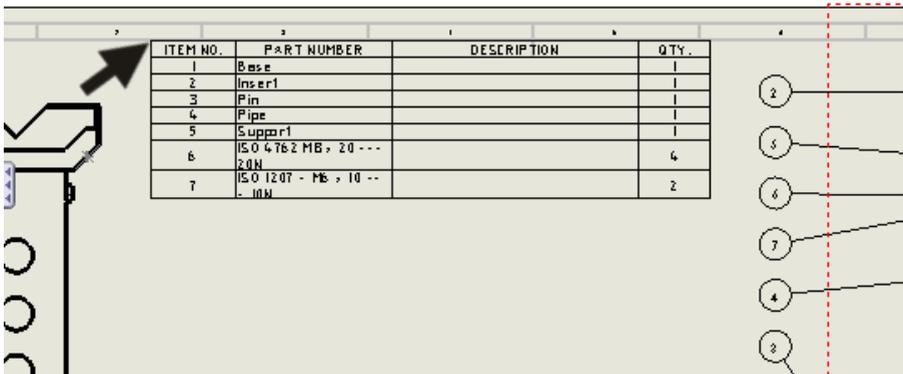
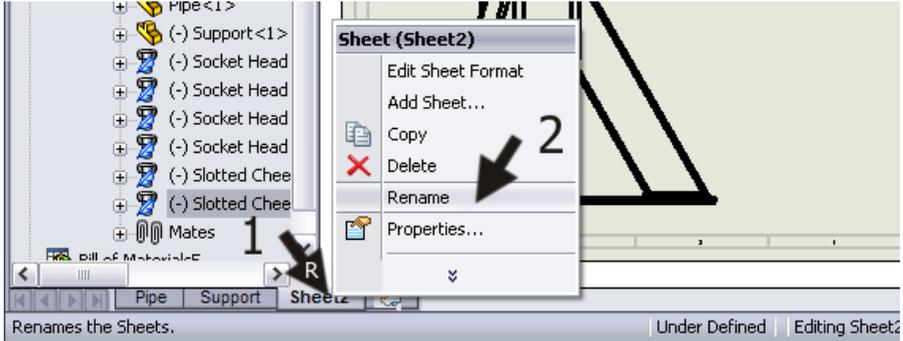
<p>71</p> <ol style="list-style-type: none"> <li>1. Click on 'View Layout' in the CommandManager.</li> <li>2. Select 'Model View'.</li> <li>3. Select the file 'Axle_stand'.</li> <li>4. Click on Next.</li> </ol>	
<p>72</p> <ol style="list-style-type: none"> <li>1. Make sure the option 'Trimetric' is checked in the PropertyManager.</li> <li>2. Put the view on the drawing sheet.</li> </ol>	
<p>73</p> <p>The exploded view may be enlarged a little.</p> <ol style="list-style-type: none"> <li>1. Select the exploded view.</li> <li>2. Change the 'Scale' to 'User Defined' in the PropertyManager.</li> <li>3. Set the 'Scale' to '1:4'.</li> </ol>	

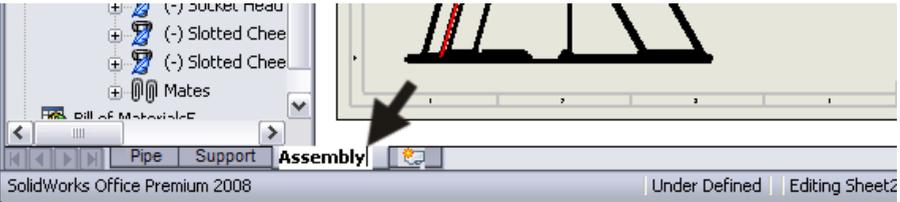
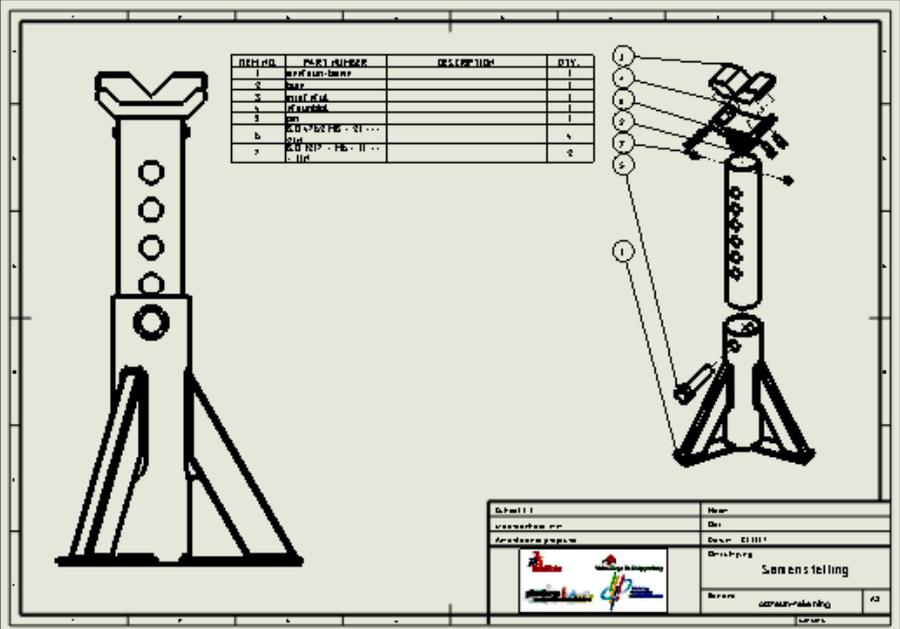
<p>74</p> <p>Next, we will put the front view on the sheet.</p> <ol style="list-style-type: none"> <li>1. Click once more on <b>'Model View'</b> in the <b>CommandManager</b>.</li> <li>2. Select the file <b>'Axle_stand'</b>.</li> <li>3. Click on Next.</li> </ol>	
<p>75</p> <ol style="list-style-type: none"> <li>1. Select the <b>Front view</b> in the <b>PropertyManager</b>.</li> <li>2. Put the view on the drawing sheet.</li> </ol> <p>Automatically, the command <b>Projected View</b> will start up. Click on OK to end this command.</p>	

<p>76</p>	<p>The front view is still exploded, but this is not meant to be. To change this, right-click on the View and select 'Properties...':</p>	
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<p>77</p>	<p>Uncheck the option 'Show in exploded state' in the menu that appears. Next, click on OK.</p>	
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<p>78</p>	<p>Change the 'Scale' to '1:2', as you have done in Step 73.</p>	
<p>79</p>	<p>We will set the part numbers to the exploded view.</p> <ol style="list-style-type: none"> <li>1. Select the exploded view.</li> <li>2. Click on 'Annotate' in the CommandManager.</li> <li>3. Click on 'AutoBalloon'.</li> </ol> <p>The part numbers now appear around the exploded view.</p>	
<p>80</p>	<ol style="list-style-type: none"> <li>1. Click on the option 'Left' in the PropertyManager to set all part numbers at the left side of the exploded view.</li> <li>2. You can drag the numbers to put them on the right if you want to.</li> <li>3. Click on OK.</li> </ol>	

<p>81</p> <p>Finally, we will add a parts list.</p> <ol style="list-style-type: none"> <li>1. Select the exploded view first.</li> <li>2. Click on 'Tables' in the CommandManager.</li> <li>3. Select 'Bill of Materials'.</li> </ol>																																	
<p>82</p> <p>Click on OK in the PropertyManager.</p>																																	
<p>83</p> <p>Put the parts list on the drawing sheet and click OK in the PropertyManager.</p>	 <table border="1" data-bbox="746 1093 1305 1256"> <thead> <tr> <th>ITEM NO.</th> <th>PART NUMBER</th> <th>DESCRIPTION</th> <th>QTY.</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>Base</td> <td></td> <td>1</td> </tr> <tr> <td>2</td> <td>Insert1</td> <td></td> <td>1</td> </tr> <tr> <td>3</td> <td>Pin</td> <td></td> <td>1</td> </tr> <tr> <td>4</td> <td>Pipe</td> <td></td> <td>1</td> </tr> <tr> <td>5</td> <td>Support1</td> <td></td> <td>1</td> </tr> <tr> <td>6</td> <td>ISO 4762 MB, 20 ...</td> <td></td> <td>4</td> </tr> <tr> <td>7</td> <td>ISO 1207 - Mb, 10 ...</td> <td></td> <td>2</td> </tr> </tbody> </table>	ITEM NO.	PART NUMBER	DESCRIPTION	QTY.	1	Base		1	2	Insert1		1	3	Pin		1	4	Pipe		1	5	Support1		1	6	ISO 4762 MB, 20 ...		4	7	ISO 1207 - Mb, 10 ...		2
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<p>84</p> <p>We must give our assembly drawing a name now. It is still called 'Sheet2' (or another number).</p> <p>Right-click on the tab which contains the composition drawing.</p> <p>Select the option 'Rename' in the menu.</p>																																	

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.	 <table border="1" data-bbox="831 593 1193 705"> <thead> <tr> <th>REF ID</th> <th>PART NUMBER</th> <th>DESCRIPTION</th> <th>QTY.</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>axle support</td> <td></td> <td>1</td> </tr> <tr> <td>2</td> <td>nut</td> <td></td> <td>1</td> </tr> <tr> <td>3</td> <td>washer</td> <td></td> <td>1</td> </tr> <tr> <td>4</td> <td>washer</td> <td></td> <td>1</td> </tr> <tr> <td>5</td> <td>nut</td> <td></td> <td>1</td> </tr> <tr> <td>6</td> <td>axle support - 21 ...</td> <td></td> <td>1</td> </tr> <tr> <td>7</td> <td>nut</td> <td></td> <td>1</td> </tr> <tr> <td>8</td> <td>axle support - 21 ...</td> <td></td> <td>1</td> </tr> </tbody> </table>	REF ID	PART NUMBER	DESCRIPTION	QTY.	1	axle support		1	2	nut		1	3	washer		1	4	washer		1	5	nut		1	6	axle support - 21 ...		1	7	nut		1	8	axle support - 21 ...		1
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	What are the main features you have learned in this tutorial?	<p>In this tutorial you have made three drawings and learned the most important functions for making a drawing. You have:</p> <ul style="list-style-type: none"> <li>- Placed views onto a drawing sheet.</li> <li>- Set dimensions in drawings, both automatically and manually.</li> <li>- Made some cross-cuts, including complete and partial cross-cuts.</li> <li>- Made an exploded view.</li> <li>- Added part numbers and a parts list to a composition drawing.</li> </ul> <p>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!</p>																																				