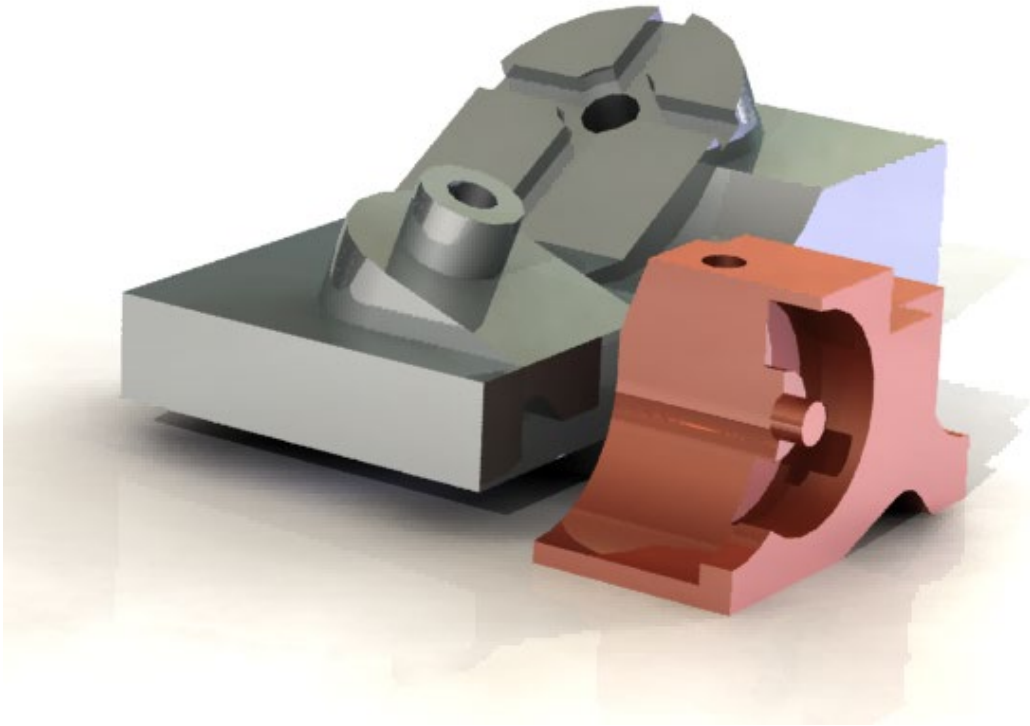


# SOLIDWORKS® tutorial 12

## CSWA



Prepatory and Advanced Vocational Training



To be used with SOLIDWORKS® Educational Release 2018-2019

© 1995-2015, Dassault Systemes SolidWorks Corporation, a Dassault Systèmes SE company, 175 Wyman Street, Waltham, Mass. 02451 USA. All Rights Reserved.

The information and the software discussed in this document are subject to change without notice and are not commitments by Dassault Systemes SolidWorks Corporation (DS SolidWorks).

No material may be reproduced or transmitted in any form or by any means, electronically or manually, for any purpose without the express written permission of DS SolidWorks.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of the license. All warranties given by DS SolidWorks as to the software and documentation are set forth in the license agreement, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of any terms, including warranties, in the license agreement.

#### **Patent Notices**

SOLIDWORKS® 3D mechanical CAD and/or Simulation software is protected by U.S. Patents 5,815,154; 6,219,049; 6,219,055; 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,079,990; 7,477,262; 7,558,705; 7,571,079; 7,590,497; 7,643,027; 7,672,822; 7,688,318; 7,694,238; 7,853,940; 8,305,376; 8,581,902; 8,817,028, 8,910,078, 9,129,083, 9,153,072 and foreign patents, (e.g., EP 1,116,190 B1 and JP 3,517,643).

eDrawings® software is protected by U.S. Patent 7,184,044; U.S. Patent 7,502,027; and Canadian Patent 2,318,706.

U.S. and foreign patents pending.

#### **Trademarks and Product Names for SOLIDWORKS Products and Services**

SOLIDWORKS, 3D ContentCentral, 3D PartStream.NET, eDrawings, and the eDrawings logo are registered trademarks and FeatureManager is a jointly owned registered trademark of DS SolidWorks.

CircuitWorks, FloXpress, PhotoView 360, and ToAnalyst are trademarks of DS SolidWorks.

FeatureWorks is a registered trademark of Geometric Ltd.

SOLIDWORKS 2018, SOLIDWORKS Standard, SOLIDWORKS Professional, SOLIDWORKS Premium, SOLIDWORKS PDM Professional, SOLIDWORKS PDM Standard, SOLIDWORKS Workgroup PDM, SOLIDWORKS Simulation, SOLIDWORKS Flow Simulation, eDrawings, eDrawings Professional, SOLIDWORKS Sustainability, SOLIDWORKS Plastics, SOLIDWORKS Electrical, SOLIDWORKS Composer, and SOLIDWORKS MBD are product names of DS SolidWorks.

Other brand or product names are trademarks or registered trademarks of their respective holders.

#### **COMMERCIAL COMPUTER SOFTWARE - PROPRIETARY**

The Software is a "commercial item" as that term is defined at 48 C.F.R. 2.101 (OCT 1995), consisting of "commercial computer software" and "commercial software documentation" as such terms are used in 48 C.F.R. 12.212 (SEPT 1995) and is provided to the U.S. Government (a) for acquisition by or on behalf of

#### **Copyright Notices for SOLIDWORKS Standard, Premium, Professional, and Education Products**

Portions of this software © 1986-2015 Siemens Product Lifecycle Management Software Inc. All rights reserved.

This work contains the following software owned by Siemens Industry Software Limited:

D-Cubed™ 2D DCM © 2015. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed™ 3D DCM © 2015. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed™ PGM © 2015. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed™ CDM © 2015. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed™ AEM © 2015. Siemens Industry Software Limited. All Rights Reserved.

Portions of this software © 1998-2015 Geometric Ltd.

Portions of this software incorporate PhysX™ by NVIDIA 2006-2010.

Portions of this software © 2001-2015 Luxology, LLC. All rights reserved, patents pending.

Portions of this software © 2007-2015 DriveWorks Ltd.

Copyright 1984-2010 Adobe Systems Inc. and its licensors. All rights reserved. Protected by U.S. Patents 5,929,866; 5,943,063; 6,289,364; 6,563,502; 6,639,593; 6,754,382; Patents Pending. Adobe, the Adobe logo, Acrobat, the Adobe PDF logo, Distiller and Reader are registered trademarks or trademarks of Adobe Systems Inc. in the U.S. and other countries.

For more DS SolidWorks copyright information, see Help > About SOLIDWORKS.

#### **Copyright Notices for SOLIDWORKS Simulation Products**

Portions of this software © 2008 Solversoft Corporation.

PCGLSS © 1992-2014 Computational Applications and System Integration, Inc. All rights reserved.

#### **Copyright Notices for SOLIDWORKS Standard Product**

© 2011, Microsoft Corporation. All rights reserved.

#### **Copyright Notices for SOLIDWORKS PDM Professional Product**

Outside In® Viewer Technology, © 1992-2012 Oracle

© 2011, Microsoft Corporation. All rights reserved.

#### **Copyright Notices for eDrawings Products**

Portions of this software © 2000-2014 Tech Soft 3D.

Portions of this software © 1995-1998 Jean-Loup Gailly and Mark Adler.

Portions of this software © 1998-2001 3Dconnexion.

Portions of this software © 1998-2014 Open Design Alliance. All rights reserved.

Portions of this software © 1995-2012 Spatial Corporation.

The eDrawings® for Windows® software is based in part on the work of the Independent JPEG Group.

Portions of eDrawings® for iPad® copyright © 1996-1999 Silicon Graphics Systems, Inc.

Portions of eDrawings® for iPad® copyright © 2003 - 2005 Apple Computer Inc.

This tutorial is developed by SOLIDWORKS Benelux and can be used by anyone for self-training purposes of the 3D CAD-program SOLIDWORKS. **Every other use of this tutorial or parts of it is prohibited.** For questions, please contact SOLIDWORKS Benelux. Contact information is printed at the last page of this tutorial.

Initiative: Kees Kloosterboer (SOLIDWORKS Benelux)

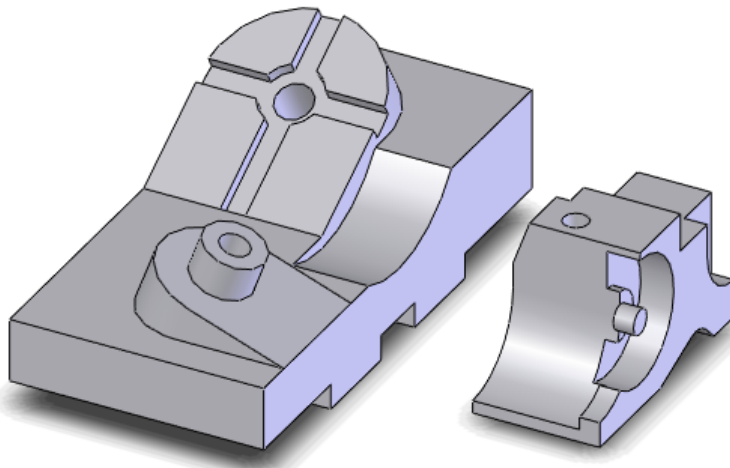
Educations advisor: Jack van den Broek

Realisation: Arnoud Breedveld (PAZ Computerworks)

## CSWA

CSWA (Certified SOLIDWORKS Associate) is a certificate you can receive after you have trained yourself in working with SOLIDWORKS. When you have to apply for a job in the future, you can simply prove that you know how to use SOLIDWORKS rather well by showing this certificate. It is certainly interesting to get it. Do you want more information about how to get this certificate, please ask your teacher for it.

Did you do all tutorials in this array and practiced some more exercises, you should be able to get the CSWA-certificate. To get more acquainted to the questions from a CSWA-test, we will practice two exercises from the CSWA test. You will not get any new topics, but you will find out how to make such an exercise.



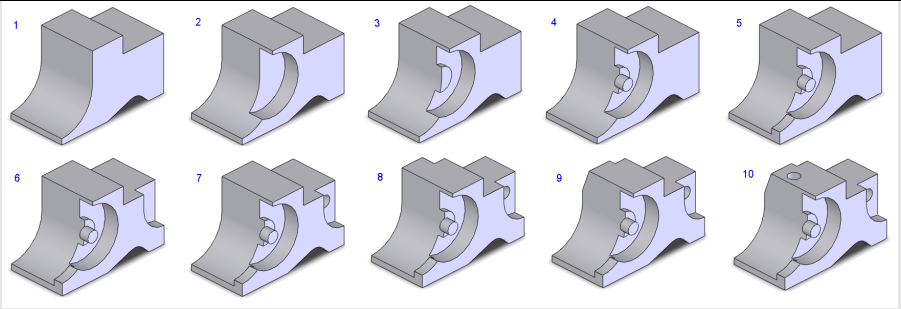
### Assignment

Available time: 45 min.

We will show you the assignment as described in the test

Build this part in SOLIDWORKS.	Your assignment is: build a part in SOLIDWORKS.
Unit system: MMGS (millimeter, gram, second)	Because inches are often used in the USA it says here that we work in millimeters and grams (metrical system). This is default in Europe, except for in England.
Decimal places: 2	We will work with two decimals: this is a default setting too.
Part Origin: Arbitrary	The Origin is at a random position, although in some assignments the position of the origin is determined.
A=63mm, B=50mm, C=100mm	Some dimensions are indicated in the model with the letters A, B or C. You will replace them with the values as given on the left.
All holes through all, unless other-	All holes will go through the whole

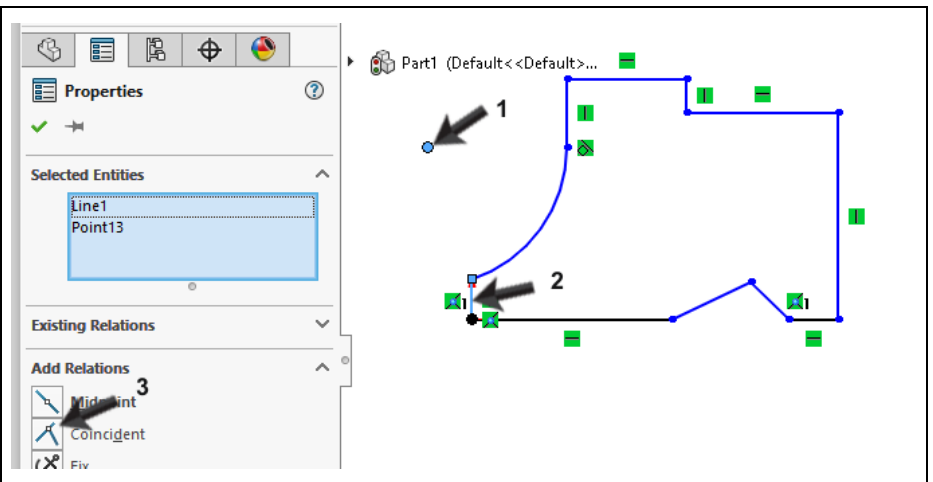
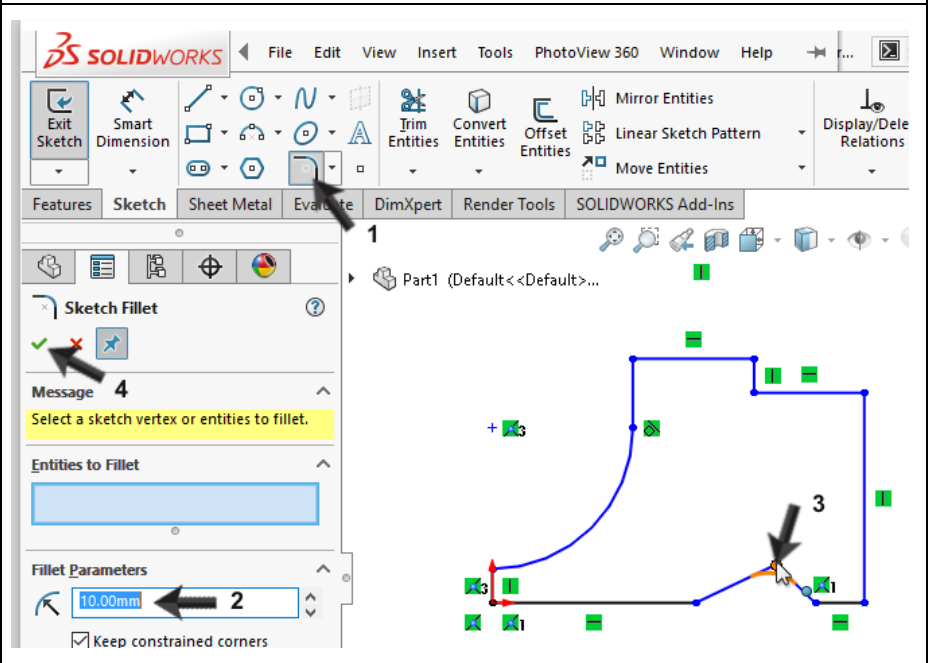
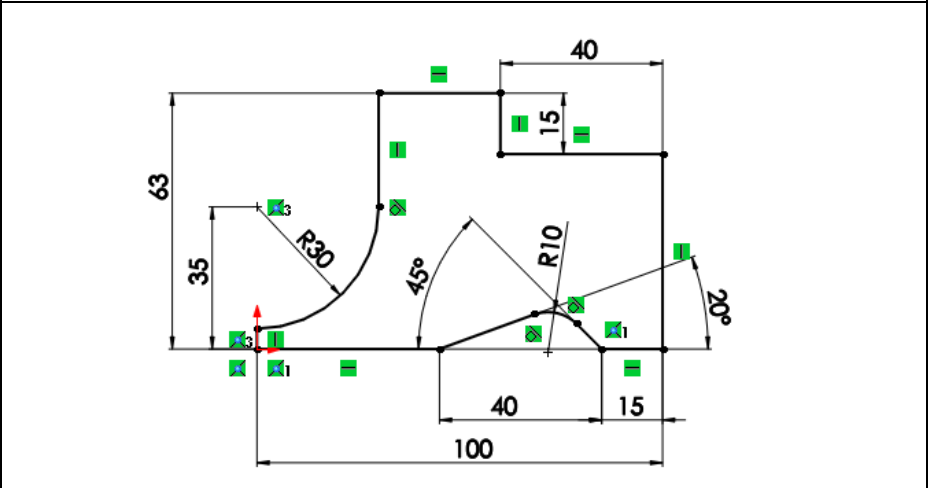


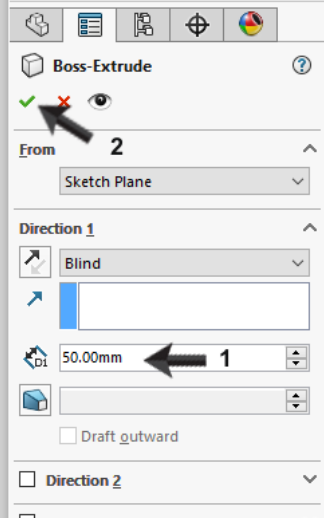
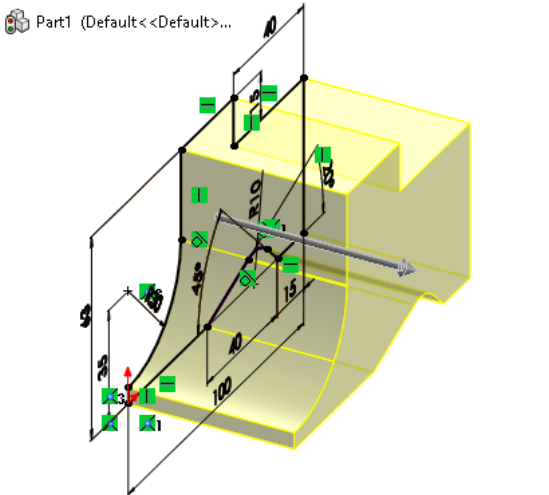
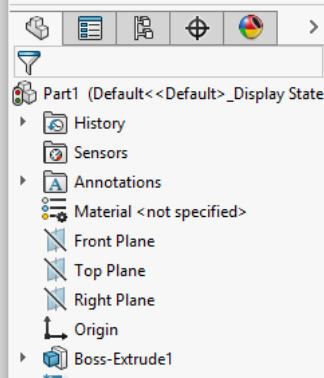
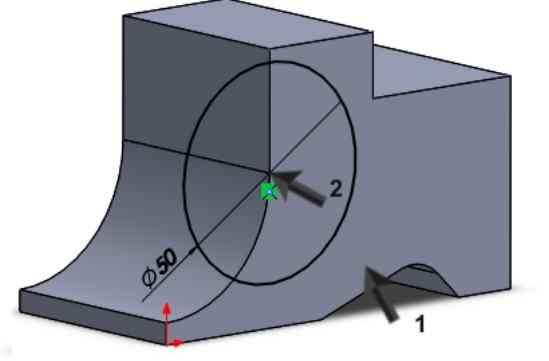
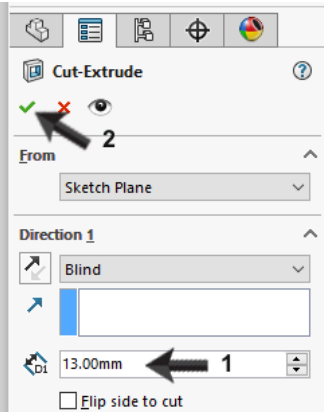
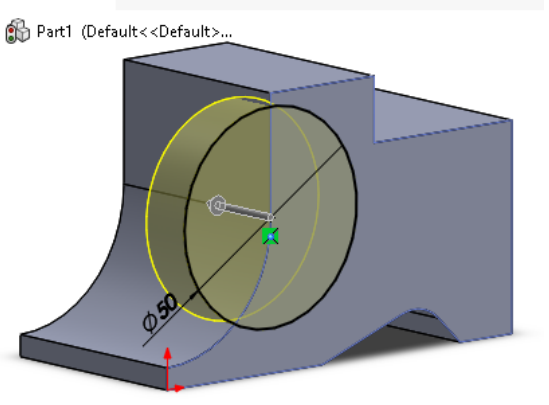
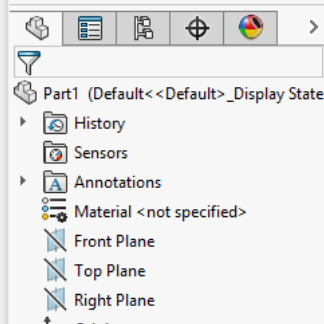
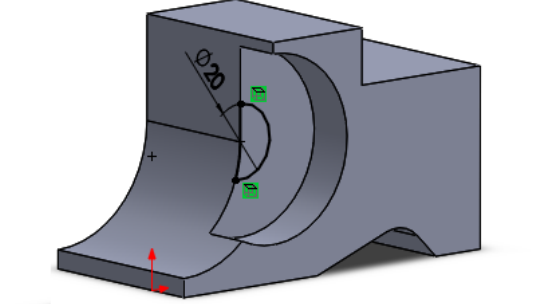


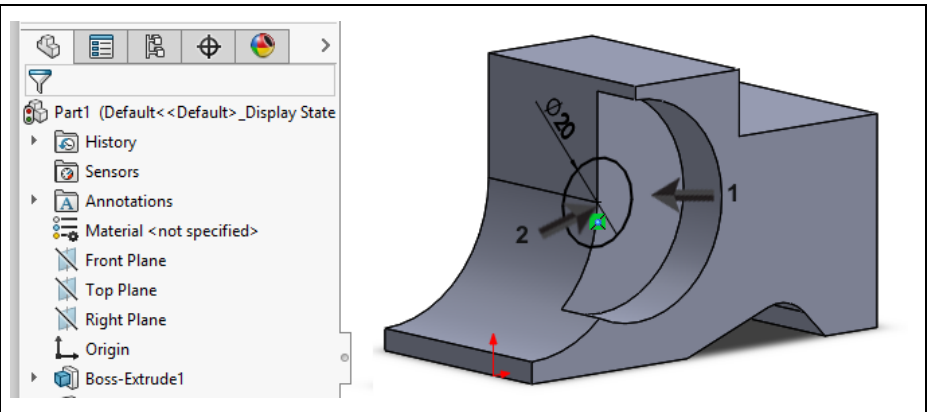
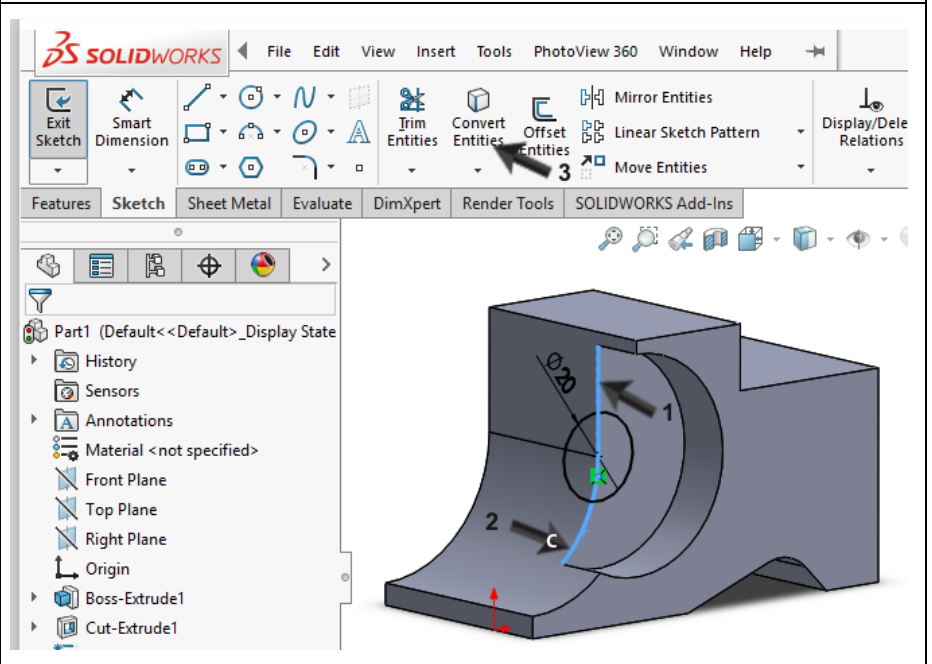
You can build your model in another way of course. There is not only one right way, but you have to do it as simple as possible, using as little features as possible.

Let us have a look how to build the model from here.

<p><b>1</b></p>	<p>Start SOLIDWORKS and open a new part.</p>	
<p><b>2</b></p>	<p>Select the Right Plane and make a sketch s shown on the right.</p> <p>Can you make this sketch yourself already? Very good. Continue with step 8 now.</p> <p>If you do not succeed doing it yourself, try using the next few steps.</p>	
<p><b>3</b></p>	<p>Draw a shape as you see on the right.</p> <p>Make sure the line from the origin runs horizontally to the right and has a length of about 50 mm. With this as a base the proportions will be right.</p>	
<p><b>4</b></p>	<p>Draw the arc now:</p> <ol style="list-style-type: none"> <li>1. Click on Arc in the CommandManager.</li> <li>2. Click on Tangent Arc in the PropertyManager</li> <li>3. Click on the lower end of the vertical line as shown on the right</li> <li>4. Click on the upper end of the vertical line as shown on the right.</li> </ol>	

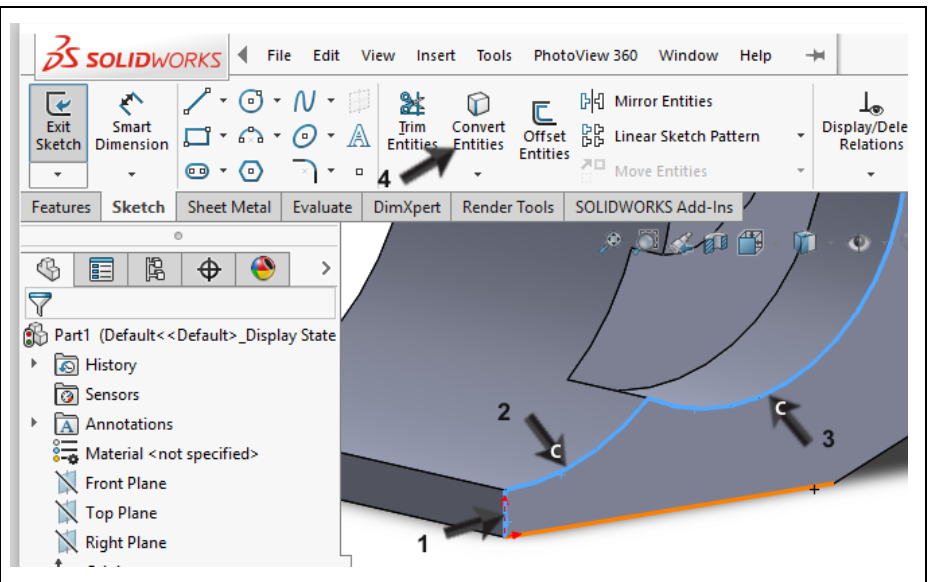
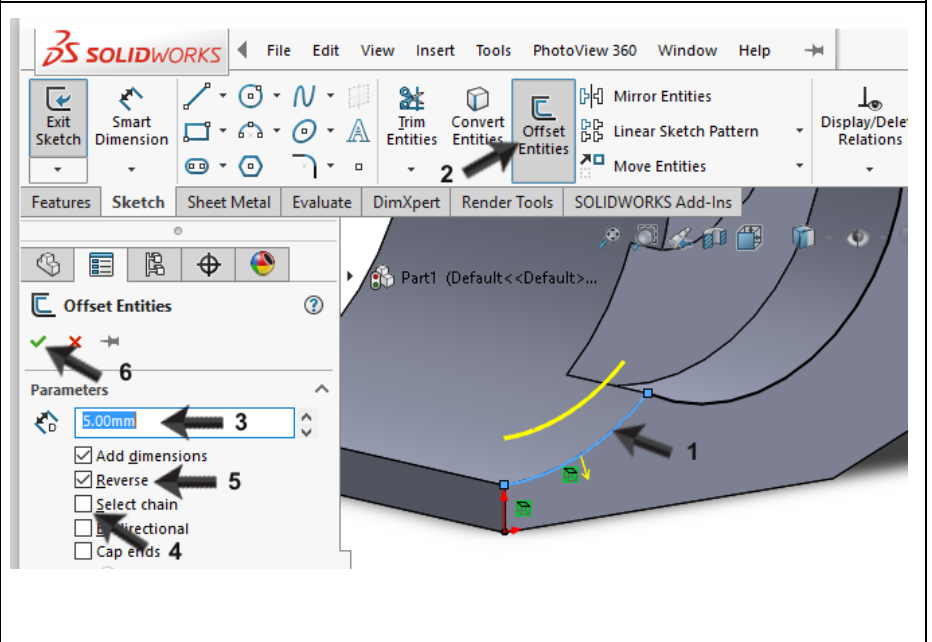
<p><b>5</b></p>	<p>Put the midpoint of the arc you have just drawn exactly on the left vertical line. By doing so, you are sure the arc is always 90°.</p> <ol style="list-style-type: none"> <li>1. Select the midpoint of the arc.</li> <li>2. Select (holding the &lt;ctrl&gt;-key) the left vertical line.</li> <li>3. Click on Coincident in the CommandManager.</li> </ol>	 <p>The screenshot shows the SolidWorks CommandManager with the 'Coincident' relation selected. The 'Selected Entities' list contains 'Line1' and 'Point13'. The 'Existing Relations' section is empty. The 'Add Relations' section shows 'Coincident' selected. The background shows a sketch of a part with a curved section. Arrows 1, 2, and 3 indicate the selection of the arc's midpoint, the vertical line, and the 'Coincident' relation respectively.</p>
<p><b>6</b></p>	<p>Make a fillet at the bottom of the sketch:</p> <ol style="list-style-type: none"> <li>1. Click on Sketch Fillet in the CommandManager</li> <li>2. Check if you have set a radius of 10 mm. in the PropertyManager (this is the default value)</li> <li>3. Click on the corner you want to fillet in the sketch.</li> </ol>	 <p>The screenshot shows the SolidWorks CommandManager with the 'Sketch Fillet' command selected. The 'Message' box says 'Select a sketch vertex or entities to fillet.' The 'Entities to Fillet' list is empty. The 'Fillet Parameters' section shows a radius of '10.00mm'. The background shows the same sketch as in step 5, but with a fillet applied to the bottom corner. Arrows 1, 2, and 3 indicate the selection of the 'Sketch Fillet' command, the radius value, and the corner to be filleted respectively.</p>
<p><b>7</b></p>	<p>Add the dimensions in the sketch as shown on the right.</p>	 <p>The dimensioned sketch shows a part with the following dimensions: a total width of 100, a total height of 63, a top horizontal edge of 40, a vertical edge of 15, a bottom horizontal edge of 40, and a bottom-right corner with a 20° angle. The sketch also features a 45° angle, a fillet with a radius of R10, and another fillet with a radius of R30. The sketch is shown in a perspective view with green selection handles.</p>

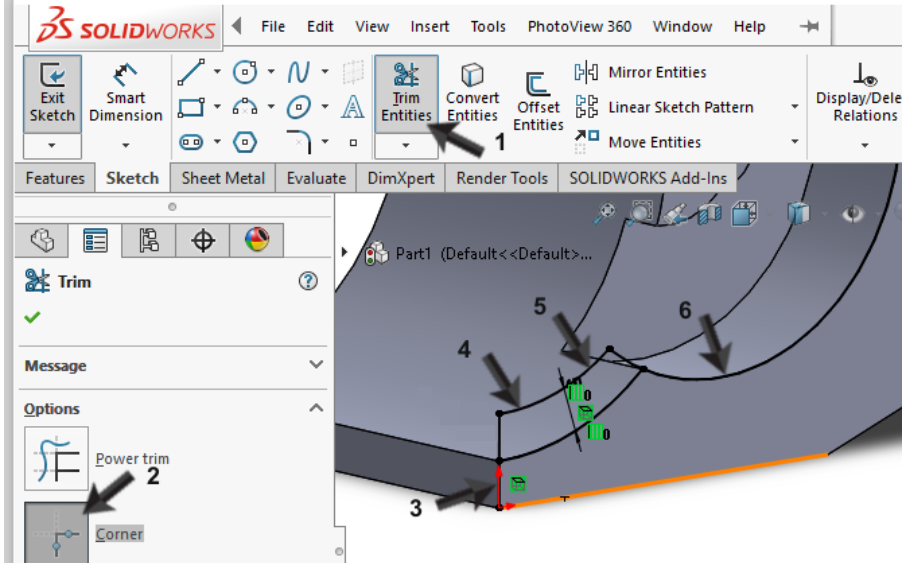
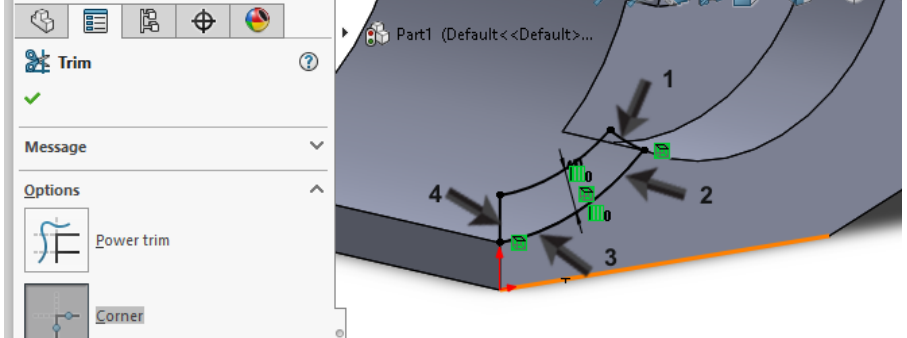
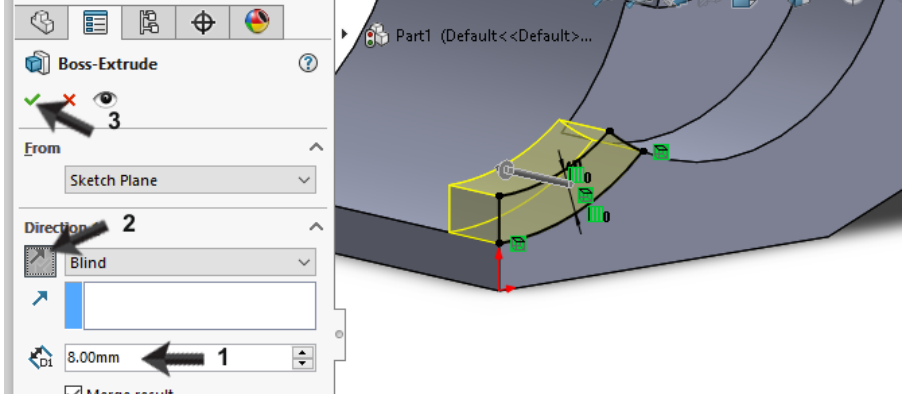
<p><b>8</b></p>	<p>Extrude the sketch over 50mm.</p>		
<p><b>9</b></p>	<p>Next make a sketch, as shown on the right.</p> <ol style="list-style-type: none"> <li>1. Select the front surface of the model to draw a new sketch on it</li> <li>2. Click on the point where the line converts into the arc.</li> </ol> <p>Draw the circle and add the dimension to the sketch.</p>		
<p><b>10</b></p>	<p>Make an Extruded Cut from the sketch, setting the depth to 13mm.</p>		
<p><b>11</b></p>	<p>Make a sketch as shown on the right.</p> <p>Can you do it yourself? Proceed to step 15.</p> <p>If this does not work out, watch the following steps which tell you how to handle this.</p>		

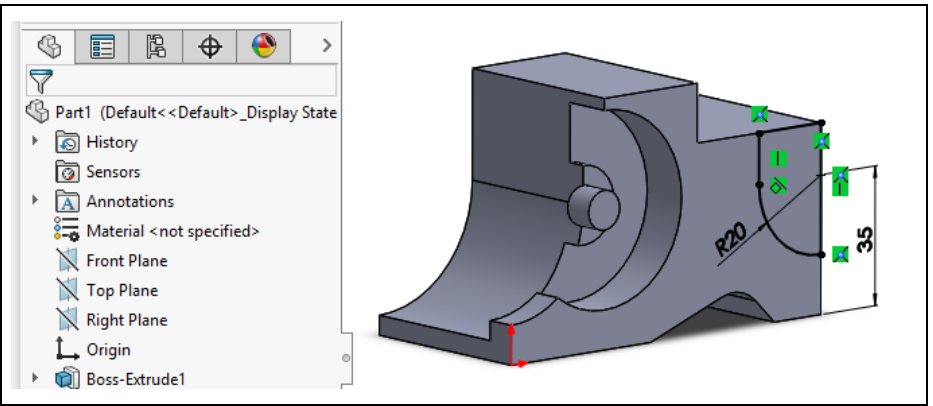
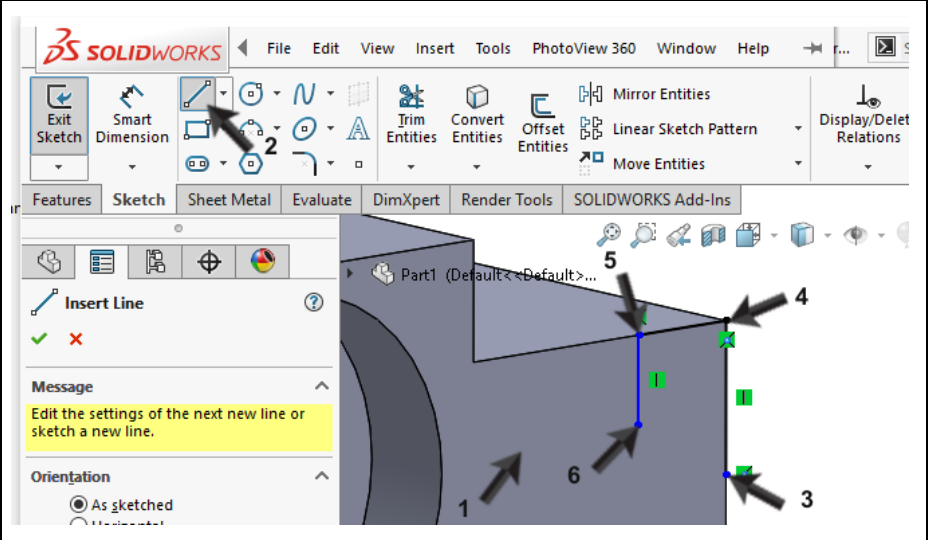
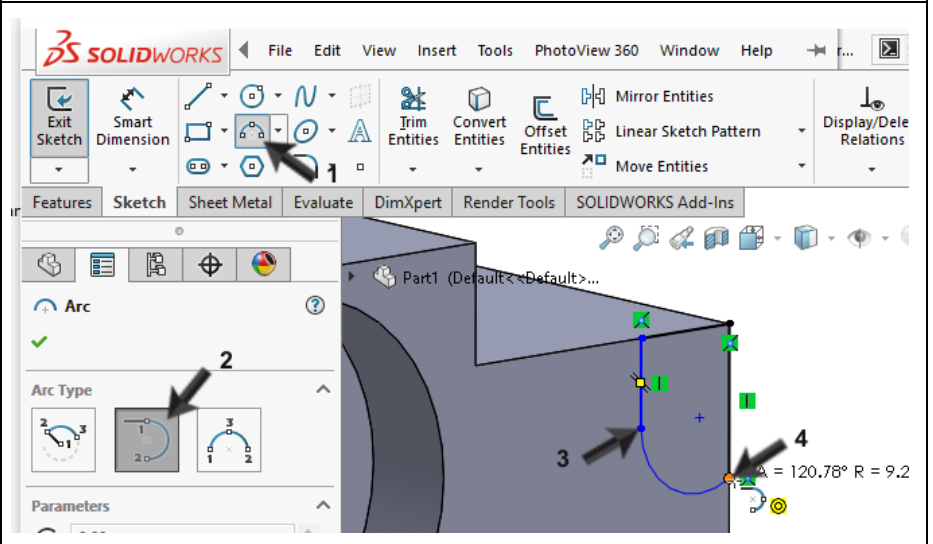
<p><b>12</b></p>	<ol style="list-style-type: none"> <li>1. Select the deeper plane first. On this surface we will make a new sketch.</li> <li>2. Draw a circle and make sure the midpoint is exactly at the point where the straight line converts in to an arc.</li> <li>3. Set the size of the circle to <math>\varnothing 20\text{mm}</math>.</li> </ol>	
<p><b>13</b></p>	<p>Push &lt;esc&gt; on your keyboard to end the Smart Dimensions-command.</p> <ol style="list-style-type: none"> <li>1,2 Select the line and the arc as shown on the right.</li> <li>3. Click on Convert Entities in the Command-Manager</li> </ol>	

<p><b>14</b></p> <ol style="list-style-type: none"> <li>1. Click on Trim Entities in the CommandManager</li> <li>2. Click on Trim to closest in the PropertyManager</li> <li>3. Click on the three parts of the sketch that need to be removed.</li> </ol>		
<p><b>15</b></p>	<p>Extrude this sketch to a depth of 5mm.</p>	
<p><b>16</b></p> <ol style="list-style-type: none"> <li>1. Select the plane to draw a sketch on.</li> <li>2. Draw a circle. Make sure the midpoint is exactly on the point where the straight line converts into an arc.</li> <li>3. Set the size of the circle to <math>\text{\O}10\text{mm}</math>.</li> </ol>		

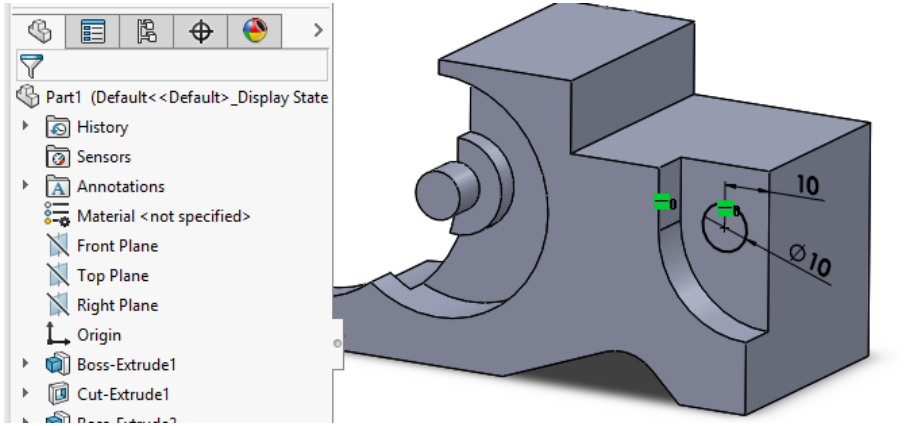
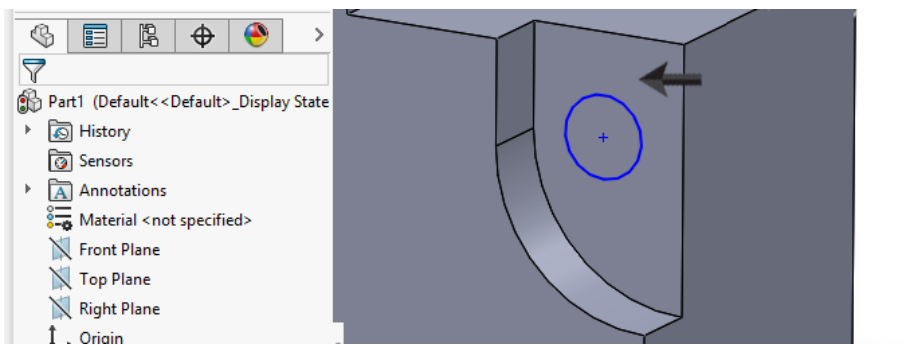
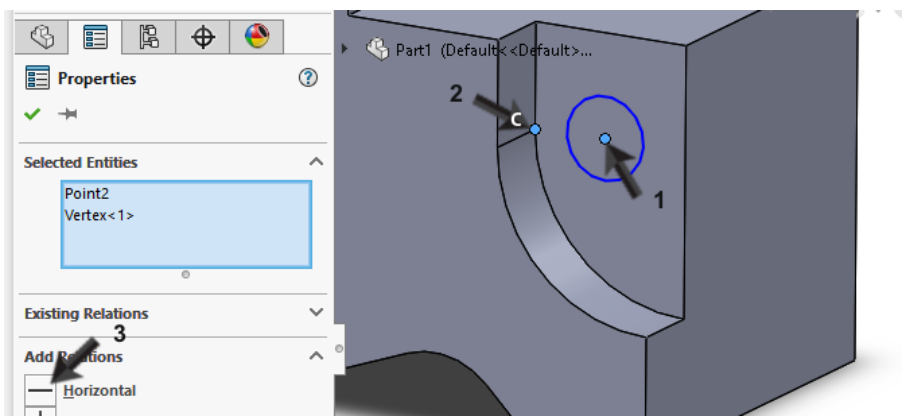
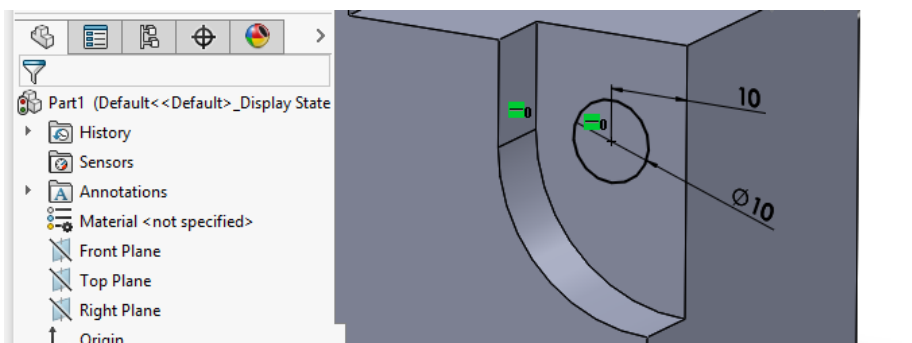
<p><b>17</b></p>	<p>Extrude the sketch to a depth of 8mm.</p>	
<p><b>18</b></p>	<p>Make the sketch as drawn on the right. Can you manage yourself? If you can, proceed to step 24. If you can not do it all by yourself, follow the next steps below.</p>	
<p><b>19</b></p>	<ol style="list-style-type: none"> <li>1. Select the plane you want to make a sketch on.</li> <li>2. Click on Sketch in the CommandManager to open the sketch.</li> </ol>	
<p><b>Tip!</b></p>	<p>Most times when we want to make a sketch we select a plane and start drawing a line or circle. SOLIDWORKS will automatically open the sketch then.</p> <p>In the last step you have opened the sketch explicitly. Why? We will use the Convert Entities command first and to be able to use this command the sketch must be open already. That is the reason for this action.</p>	

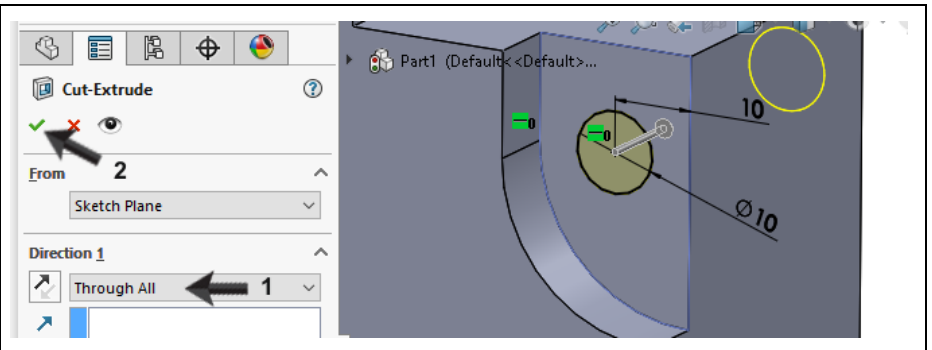
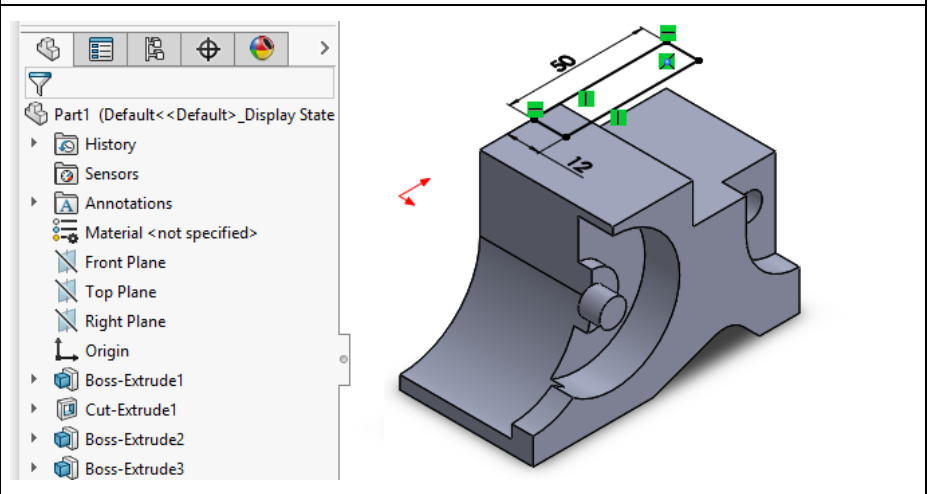
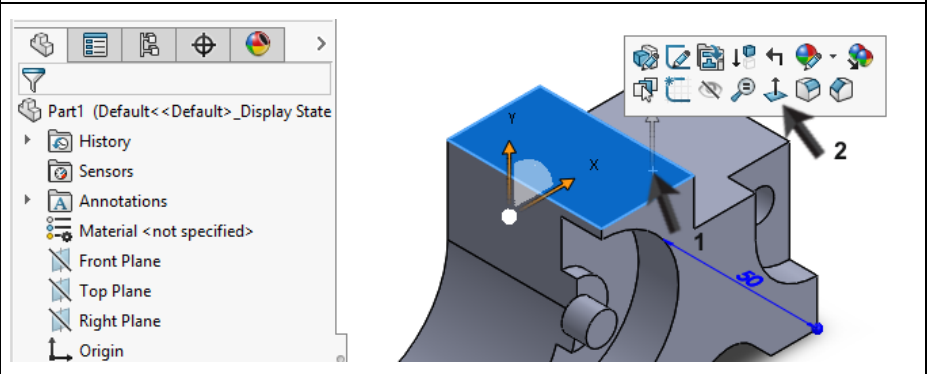
<p><b>20</b></p>	<ol style="list-style-type: none"> <li>1-3 Select the three edges in the model as shown on the right.</li> <li>4. Click on Convert Entities in the CommandManager.</li> <li>5. In the PropertyManager, click OK.</li> </ol>	
<p><b>21</b></p>	<ol style="list-style-type: none"> <li>1. Select the edge as shown in the illustration</li> <li>2. Click on Offset Entities in the CommandManager</li> <li>3. Set the distance to 5mm in the PropertyManager</li> <li>4. Uncheck the option Select Chain.</li> <li>5. Check the option Reverse to be sure the copy will be put at the right side.</li> <li>6. Click OK.</li> </ol>	

<p><b>22</b></p> <ol style="list-style-type: none"> <li>1. Click on Trim Entities in the CommandManager</li> <li>2. Select the option Corner in the Property-Manager</li> <li>3-6 Make the upper corners by clicking as indicated in the illustration.</li> </ol>	
<p><b>23</b></p> <p>Next make the bottom corner points by clicking as shown on the right.</p>	
<p><b>24</b></p> <p>Extrude this sketch over 8mm.</p> <p>Use the Reverse Direction key to make sure the extrusion extends in the right direction.</p>	

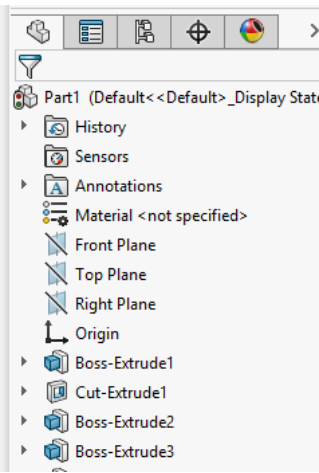
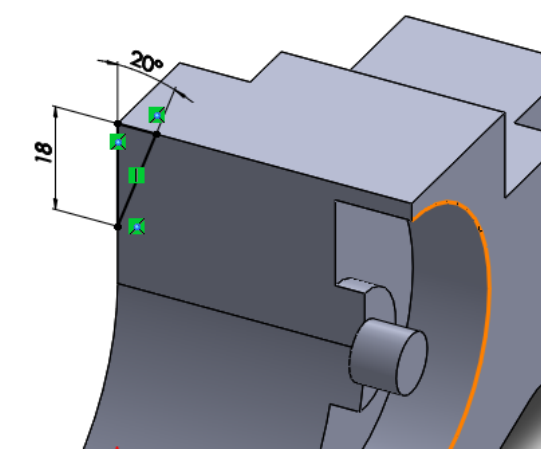
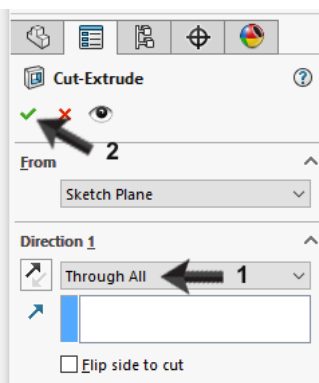
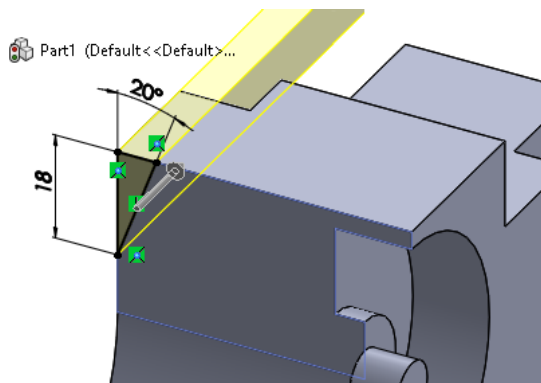
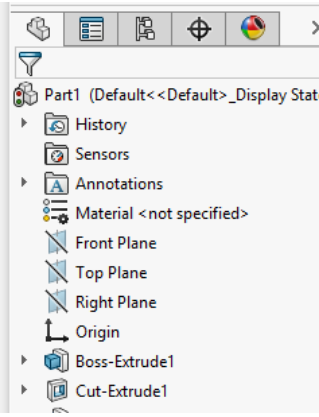
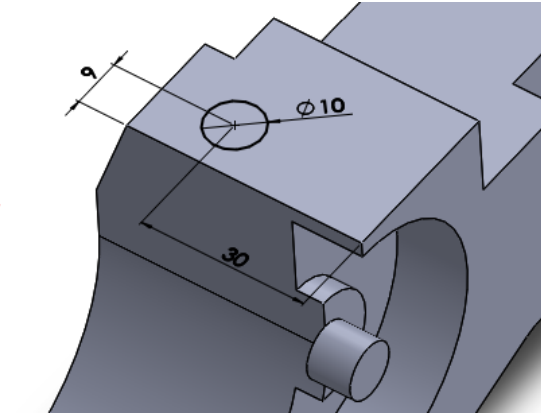
<p><b>25</b></p>	<p>Make the sketch as shown.</p> <p>Can you manage this yourself? Continue at step 30. If not, follow the next few steps.</p>	
<p><b>26</b></p>	<p>Select the plane on which you want to make a sketch.</p> <p>Draw three straight lines as shown in the illustration.</p>	
<p><b>27</b></p>	<ol style="list-style-type: none"> <li>1. Click on Tangent Arc in the CommandManager</li> <li>2. Click at the bottom end of the left vertical line.</li> <li>3. Click on the bottom end of the right vertical line.</li> </ol>	

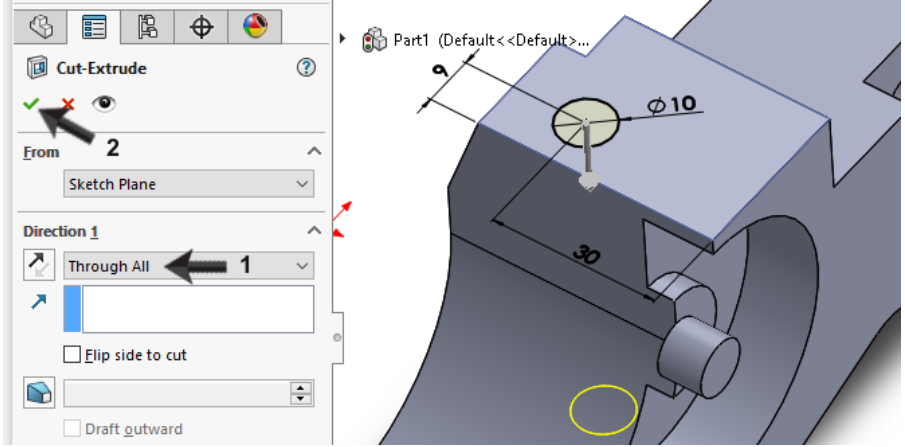
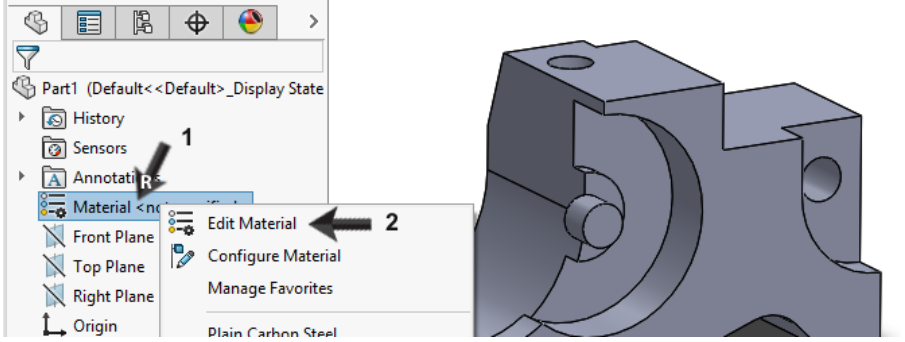
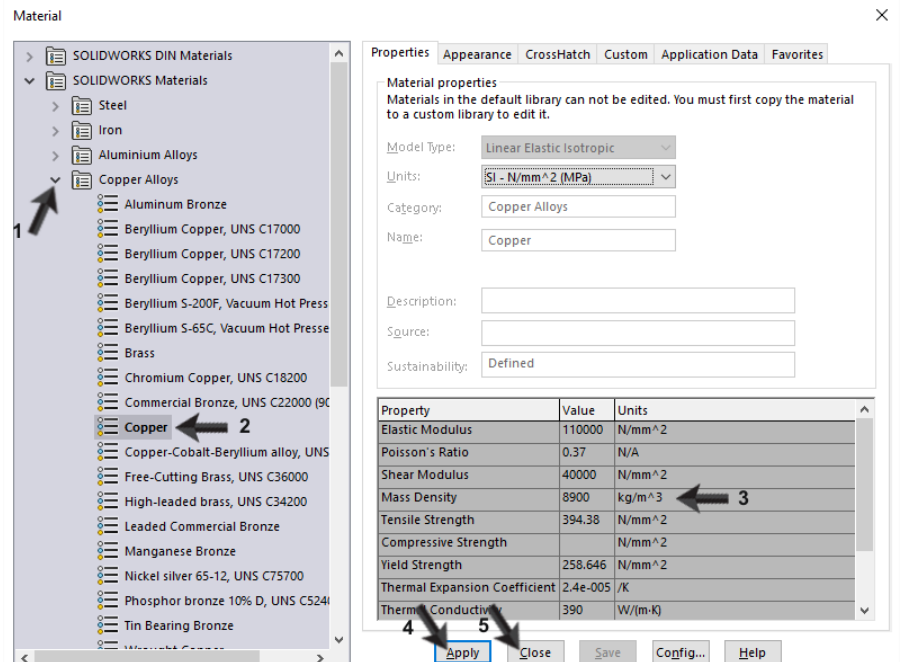
<p><b>28</b></p>	<ol style="list-style-type: none"> <li>1. Select the midpoint of the arc.</li> <li>2. Hold the &lt;ctrl&gt;-key and select the right vertical line too.</li> <li>3. Click on Coincident in the PropertyManager.</li> </ol>	
<p><b>29</b></p>	<p>Add the two dimensions as shown.</p>	
<p><b>30</b></p>	<p>Make an Extruded Cut from this sketch with a depth of 9mm.</p>	

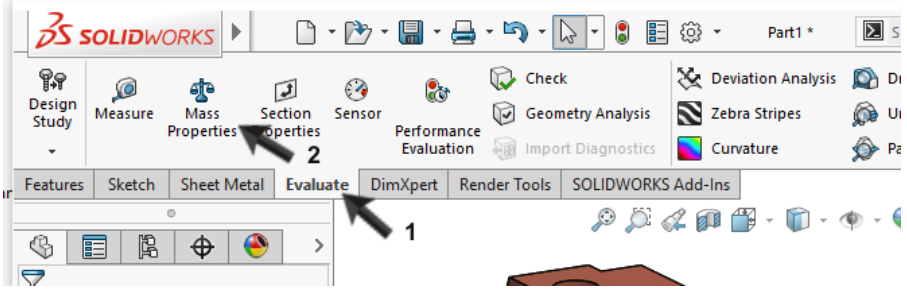
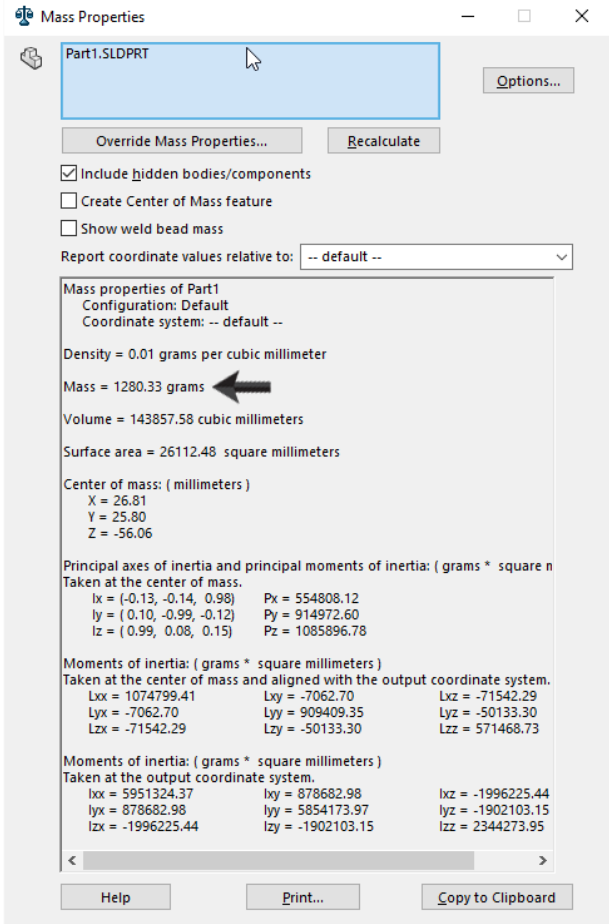
<p><b>31</b></p>	<p>Make the sketch as shown and continue to step 35.</p> <p>If you can not manage yourself, follow the next few steps.</p>	
<p><b>32</b></p>	<ol style="list-style-type: none"> <li>1. Select the plane to make the next sketch as shown on the right.</li> <li>2. Draw a circle, just about the size and position as in the illustration.</li> </ol>	
<p><b>33</b></p>	<ol style="list-style-type: none"> <li>1. Select the midpoint from the circle.</li> <li>2. Hold the &lt;ctrl&gt;-key and click on the point as shown on the right.</li> <li>3. Click on Horizontal in the PropertyManager.</li> </ol>	
<p><b>34</b></p>	<p>Set the sizes as shown in the illustration.</p>	

<p><b>35</b></p>	<p>Make an Extruded Cut from this sketch.</p> <p>Select the option Through All.</p>	
<p><b>36</b></p>	<p>Make the sketch as shown on the right and continue to step 40.</p> <p>If you can not manage yourself, follow the next few steps.</p>	
<p><b>37</b></p>	<ol style="list-style-type: none"> <li>1. Select the upper surface from the model</li> <li>2. Click on Normal To in the popup-menu.</li> </ol>	

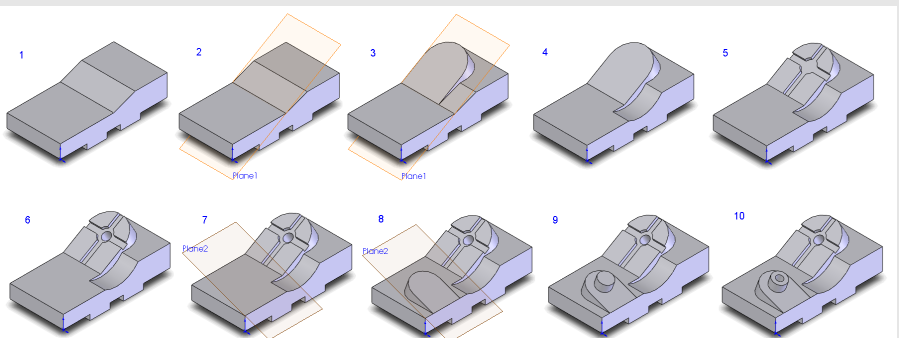
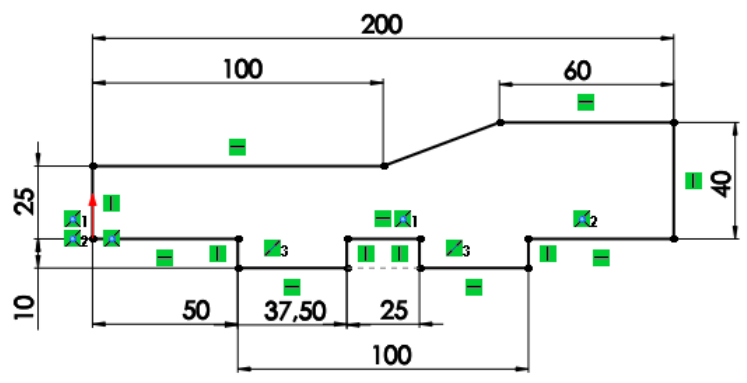
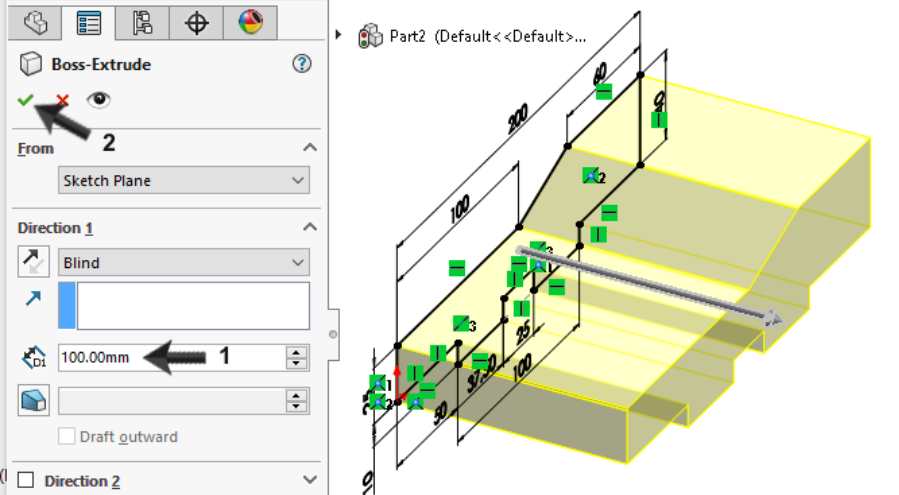
<p><b>38</b></p> <ol style="list-style-type: none"> <li>1. Click on Rectangle in the CommandManager</li> <li>2. Draw the rectangle as shown in the illustration on the right.</li> </ol>	
<p><b>39</b></p> <p>Set the two sizes as shown.</p>	
<p><b>40</b></p> <p>Make an extruded Cut from this sketch and set the depth to: Through all.</p>	

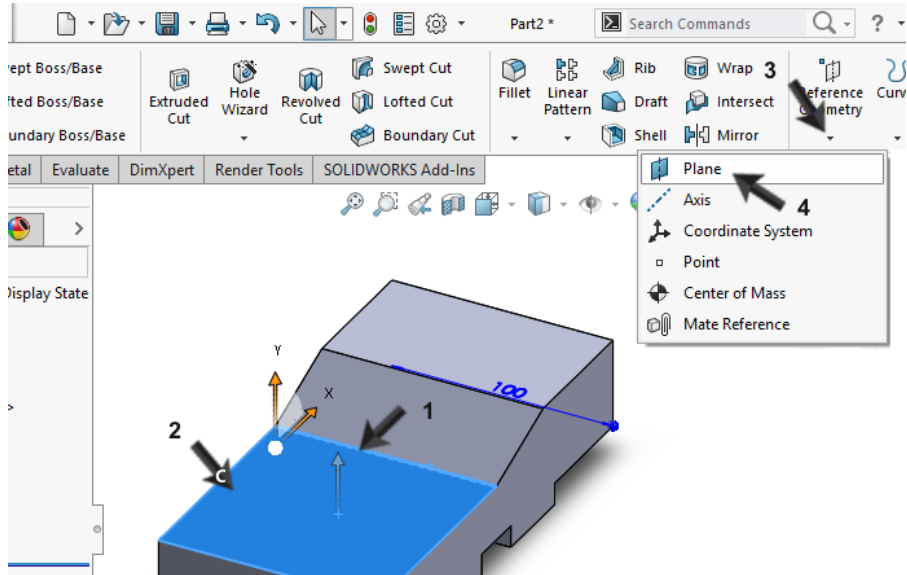
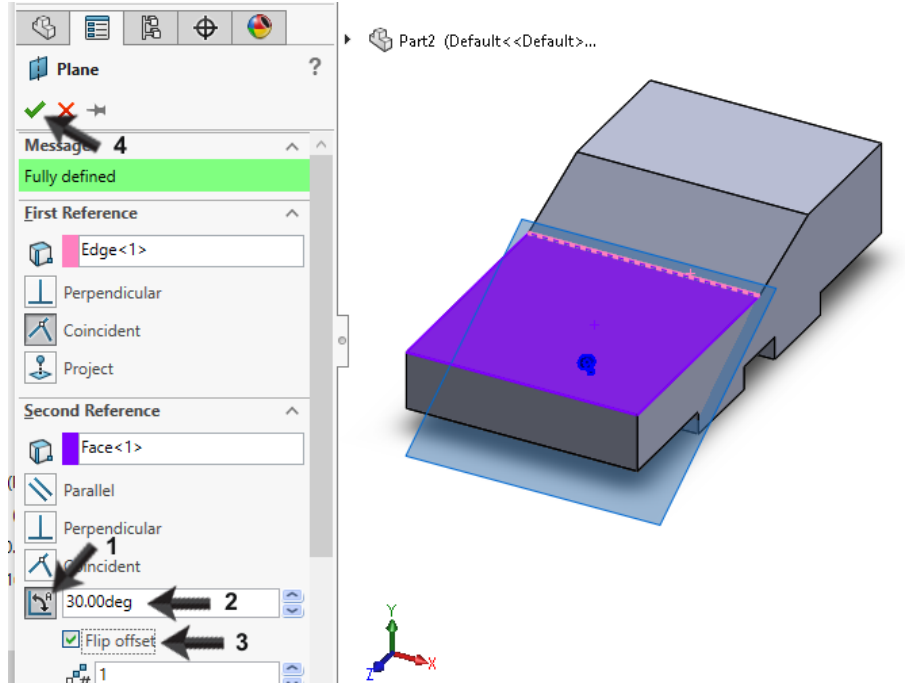
<p><b>41</b> Make the sketch as shown.</p>			
<p><b>42</b> Make an Extruded Cut from this sketch and set the depth to: Through All.</p>			
<p><b>43</b> Finally: Make the sketch as shown in the illustration on the right.</p>			

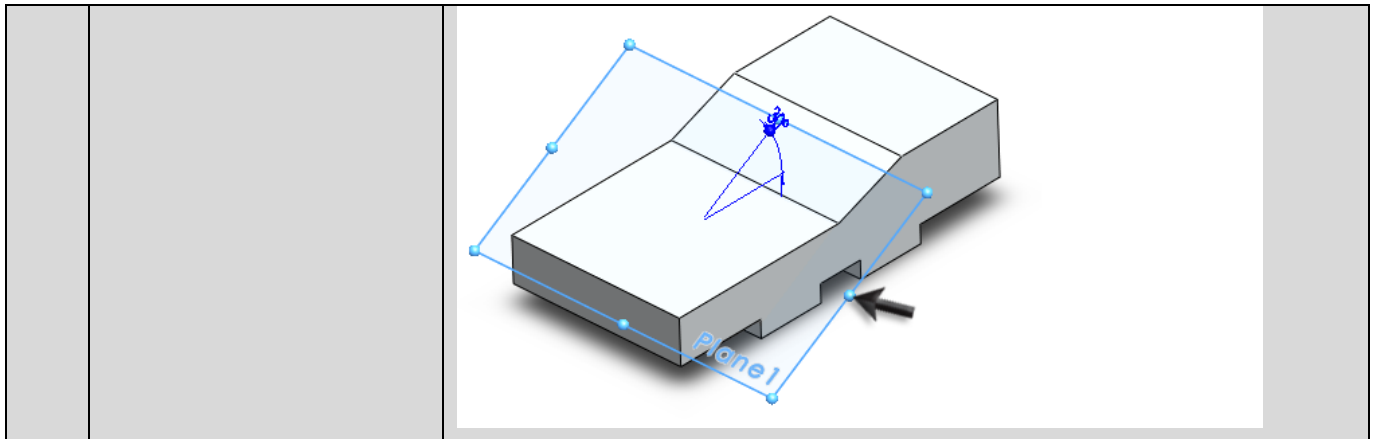
<p><b>44</b> Make an Extruded Cut from this sketch and set the depth to: Through All.</p>																															
<p><b>45</b> The model is ready now. We have to select the kind of material: the assignment says 'copper'.</p> <ol style="list-style-type: none"> <li>1. Right-click on Material in the FeatureManager.</li> <li>2. When Copper is in the list, you can click on it. If not, click on Edit Material.</li> </ol>																															
<p><b>46</b> If you chose Edit Material in the previous step, the menu will appear in which you can select a material for the model.</p> <ol style="list-style-type: none"> <li>1. Click on 'Copper Alloys'.</li> <li>2. Selecter Copper.</li> <li>3. Just to be sure: check the density at Physical Properties. Is it the same as in the assignment?</li> <li>4. Click OK.</li> <li>5. Click on Close</li> </ol>	 <table border="1" data-bbox="970 1460 1495 1697"> <thead> <tr> <th>Property</th> <th>Value</th> <th>Units</th> </tr> </thead> <tbody> <tr> <td>Elastic Modulus</td> <td>110000</td> <td>N/mm<sup>2</sup></td> </tr> <tr> <td>Poisson's Ratio</td> <td>0.37</td> <td>N/A</td> </tr> <tr> <td>Shear Modulus</td> <td>40000</td> <td>N/mm<sup>2</sup></td> </tr> <tr> <td>Mass Density</td> <td>8900</td> <td>kg/m<sup>3</sup></td> </tr> <tr> <td>Tensile Strength</td> <td>394.38</td> <td>N/mm<sup>2</sup></td> </tr> <tr> <td>Compressive Strength</td> <td></td> <td>N/mm<sup>2</sup></td> </tr> <tr> <td>Yield Strength</td> <td>258.646</td> <td>N/mm<sup>2</sup></td> </tr> <tr> <td>Thermal Expansion Coefficient</td> <td>2.4e-005</td> <td>/K</td> </tr> <tr> <td>Thermal Conductivity</td> <td>390</td> <td>W/(m·K)</td> </tr> </tbody> </table>	Property	Value	Units	Elastic Modulus	110000	N/mm <sup>2</sup>	Poisson's Ratio	0.37	N/A	Shear Modulus	40000	N/mm <sup>2</sup>	Mass Density	8900	kg/m <sup>3</sup>	Tensile Strength	394.38	N/mm <sup>2</sup>	Compressive Strength		N/mm <sup>2</sup>	Yield Strength	258.646	N/mm <sup>2</sup>	Thermal Expansion Coefficient	2.4e-005	/K	Thermal Conductivity	390	W/(m·K)
Property	Value	Units																													
Elastic Modulus	110000	N/mm <sup>2</sup>																													
Poisson's Ratio	0.37	N/A																													
Shear Modulus	40000	N/mm <sup>2</sup>																													
Mass Density	8900	kg/m <sup>3</sup>																													
Tensile Strength	394.38	N/mm <sup>2</sup>																													
Compressive Strength		N/mm <sup>2</sup>																													
Yield Strength	258.646	N/mm <sup>2</sup>																													
Thermal Expansion Coefficient	2.4e-005	/K																													
Thermal Conductivity	390	W/(m·K)																													

<p><b>47</b></p> <p>We need to know the weight of this part:</p> <ol style="list-style-type: none"> <li>1. Click on the tab Evaluate in the Command-Manager.</li> <li>2. Click on Mass Properties.</li> </ol>	
<p><b>48</b></p> <p>In the popup-menu you can read the weight: 1280.33 grams. This is answer <b>B</b> from the assignment.</p>	 <p>Part1.SLDPRT</p> <p>Options...</p> <p>Override Mass Properties... Recalculate</p> <p><input checked="" type="checkbox"/> Include hidden bodies/components</p> <p><input type="checkbox"/> Create Center of Mass feature</p> <p><input type="checkbox"/> Show weld bead mass</p> <p>Report coordinate values relative to: -- default --</p> <p>Mass properties of Part1 Configuration: Default Coordinate system: -- default --</p> <p>Density = 0.01 grams per cubic millimeter</p> <p><b>Mass = 1280.33 grams</b></p> <p>Volume = 143857.58 cubic millimeters</p> <p>Surface area = 26112.48 square millimeters</p> <p>Center of mass: ( millimeters ) X = 26.81 Y = 25.80 Z = -56.06</p> <p>Principal axes of inertia and principal moments of inertia: ( grams * square millimeters ) Taken at the center of mass. I<sub>x</sub> = (-0.13, -0.14, 0.98)    P<sub>x</sub> = 554808.12 I<sub>y</sub> = ( 0.10, -0.99, -0.12)    P<sub>y</sub> = 914972.60 I<sub>z</sub> = ( 0.99, 0.08, 0.15)    P<sub>z</sub> = 1085896.78</p> <p>Moments of inertia: ( grams * square millimeters ) Taken at the center of mass and aligned with the output coordinate system. L<sub>xx</sub> = 1074799.41    L<sub>xy</sub> = -7062.70    L<sub>xz</sub> = -71542.29 L<sub>yx</sub> = -7062.70    L<sub>yy</sub> = 909409.35    L<sub>yz</sub> = -50133.30 L<sub>zx</sub> = -71542.29    L<sub>zy</sub> = -50133.30    L<sub>zz</sub> = 571468.73</p> <p>Moments of inertia: ( grams * square millimeters ) Taken at the output coordinate system. I<sub>xx</sub> = 5951324.37    I<sub>xy</sub> = 878682.98    I<sub>xz</sub> = -1996225.44 I<sub>yx</sub> = 878682.98    I<sub>yy</sub> = 5854173.97    I<sub>yz</sub> = -1902103.15 I<sub>zx</sub> = -1996225.44    I<sub>zy</sub> = -1902103.15    I<sub>zz</sub> = 2344273.95</p> <p>Help    Print...    Copy to Clipboard</p>

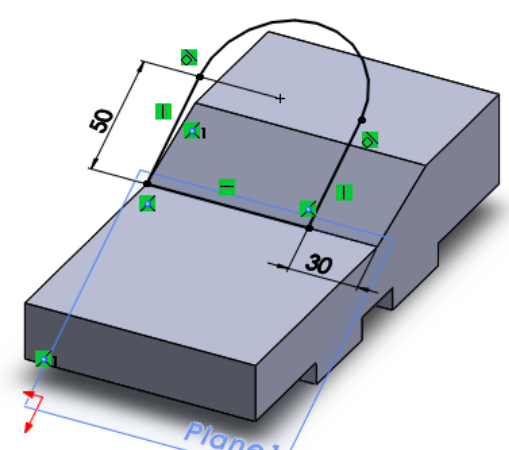
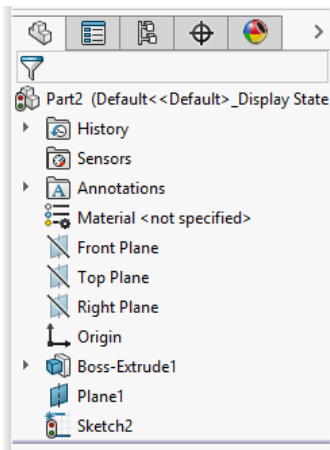


	<p><b>Work plan</b></p>	<p>Again, you have to think about the way you are going to build this model. Down below are the steps you must take. Every step is a feature.</p> 
<p><b>49</b></p>	<p>Open a new part and make the sketch as shown on the right on the Right Plane.</p>	
<p><b>50</b></p>	<p>Extrude the sketch to 100 mm</p>	

<p><b>51</b></p> <p>We will create the first reference plane:</p> <ol style="list-style-type: none"> <li>1. Select the edge as shown.</li> <li>2. Hold the &lt;ctrl&gt;-key and select the plane as shown in the illustration too.</li> <li>3. Click on the arrow beneath Reference Geometry in the CommandManager.</li> <li>4. Click on Plane.</li> </ol>	
<p><b>52</b></p> <ol style="list-style-type: none"> <li>1. In PropertyManager, click Angle.</li> <li>2. Set angle of the new plane to 30 °.</li> <li>3. Select Flip.</li> <li>4. Click ok.</li> </ol>	
<p><b>Tip!</b></p>	<p>If the edge of the reference plane is exactly collinear with the edge of the plane you selected in step 51, you'd best enlarge the reference plane a bit. This makes it easier to select it later on. Select the plane, and drag the circles that appear near the edges. By doing so, the position of the plane won't change. It will just be shown a bit larger.</p>

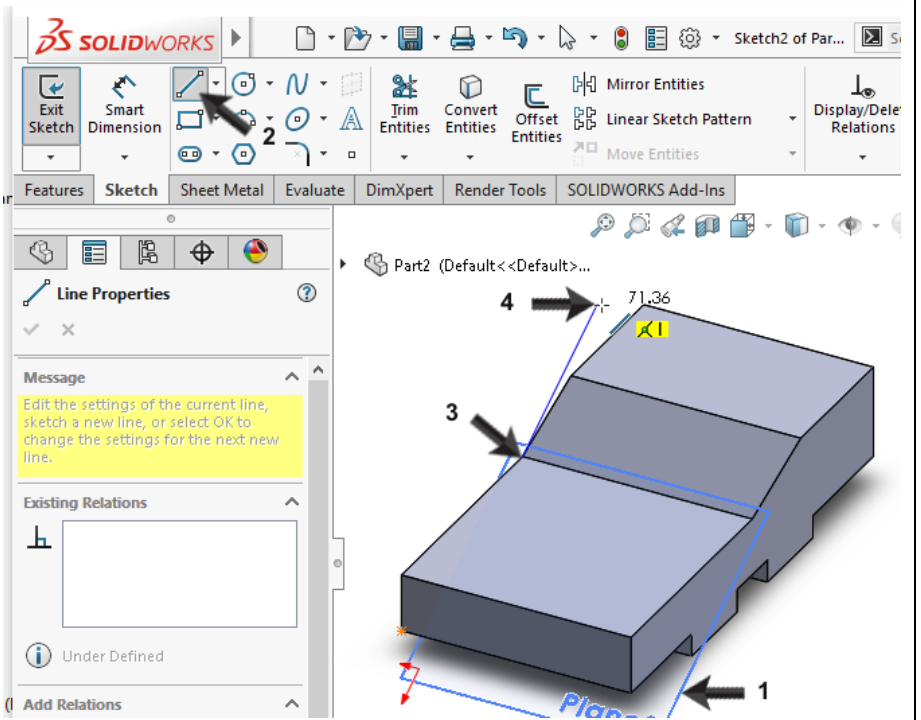


**53** Now draw the sketch that you see next to, and proceed to step 58.  
Is not it possible to make this sketch? Then follow the steps below.



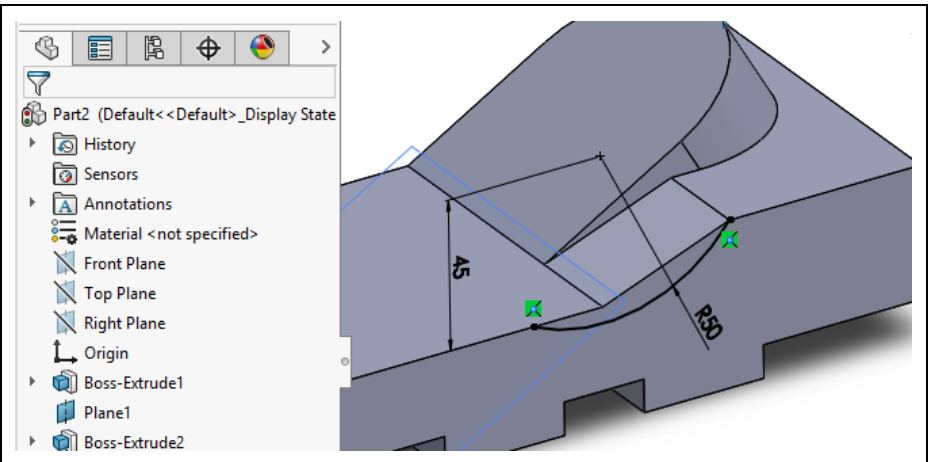
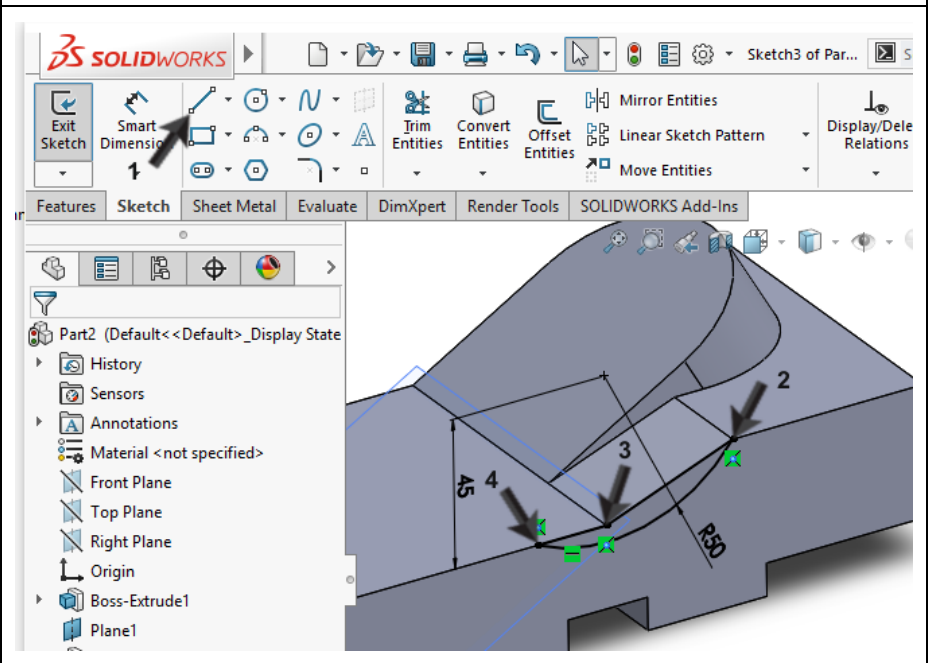
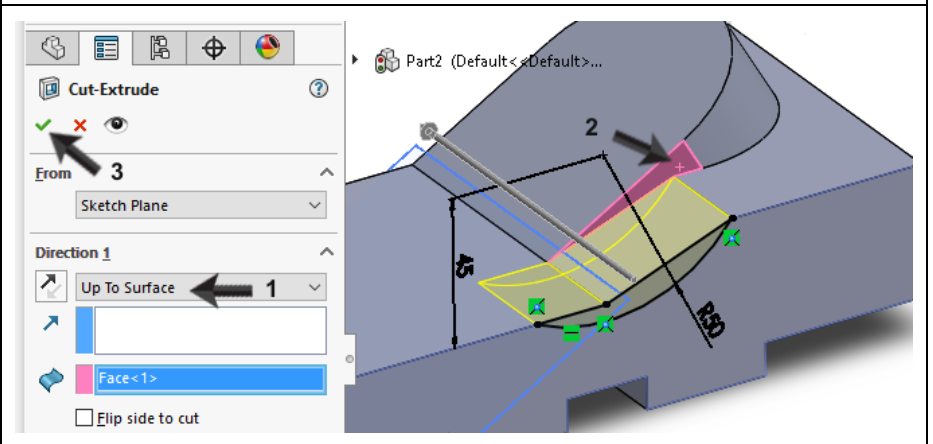
**54**

- 1 Select the help level you just created.
- 2 In the CommandManager, click Line
- 3 Click for the first point of the line as you see next to it.
- 4 Also click for the second point of the line as you see next to it.

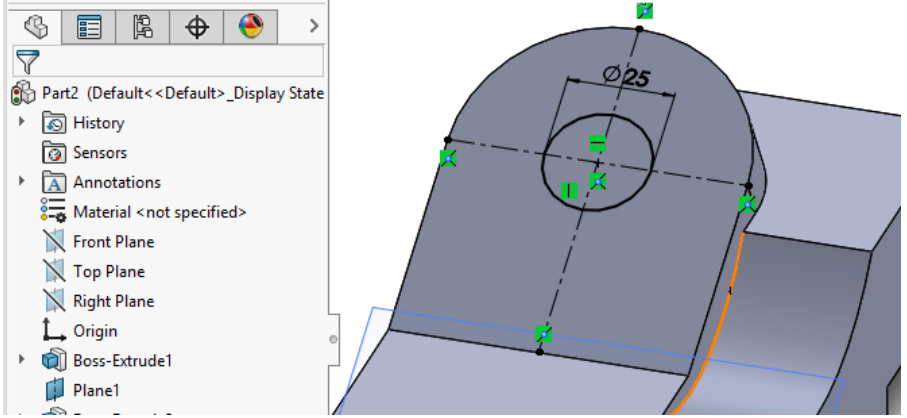
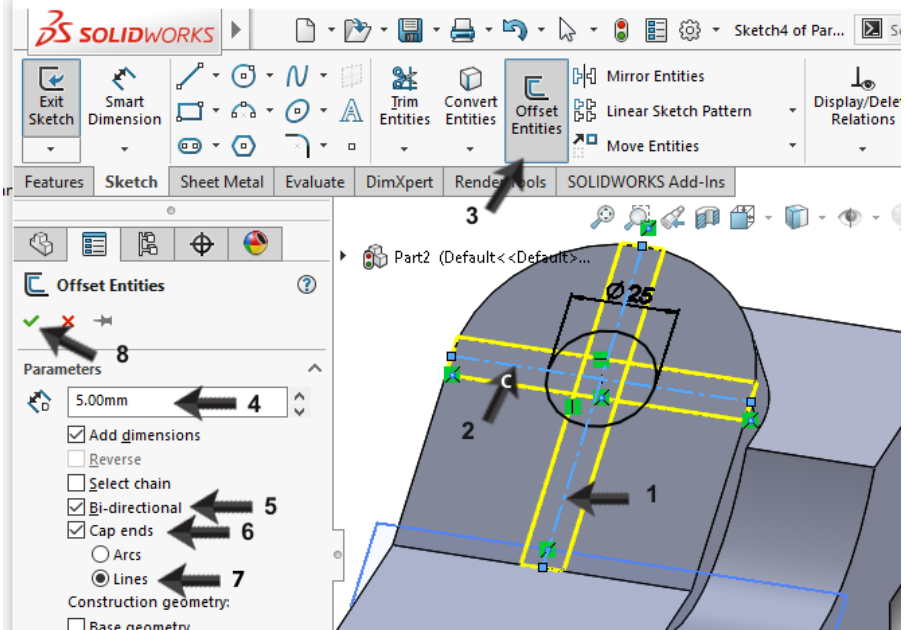


<p><b>55</b></p>	<ol style="list-style-type: none"> <li>1. Move the cursor away from the last point and then return to it (do NOT click!) SOLIDWORKS starts drawing an arc now.</li> <li>2. Click as shown to get the second point of the arc. Make sure to draw half a circle. Note the dashed construction line.</li> </ol>	
<p><b>56</b></p>	<p>SOLIDWORKS will automatically draw lines again. Draw the two last lines.</p>	
<p><b>Tip!</b></p>	<p>You saw an 'automatic' change of function between the Line and Circle command. This is called Autotransitioning in SOLIDWORKS and is very convenient if you want to build a sketch from lines and coincident circles.</p>	
<p><b>57</b></p>	<p>Add the two dimensions as shown with Smart Dimensions.</p>	

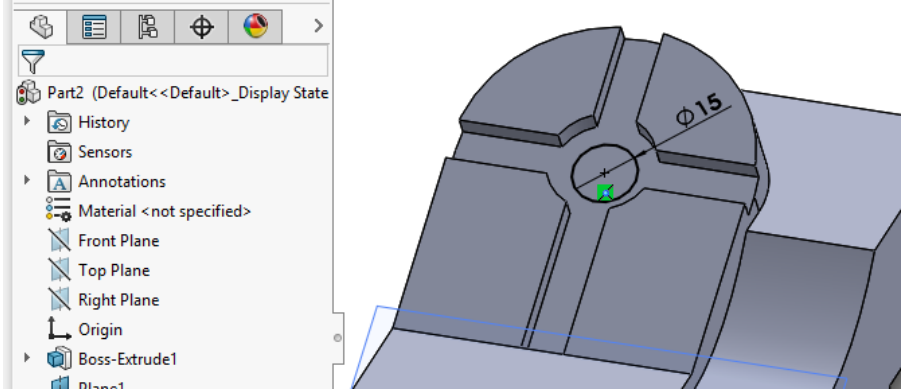
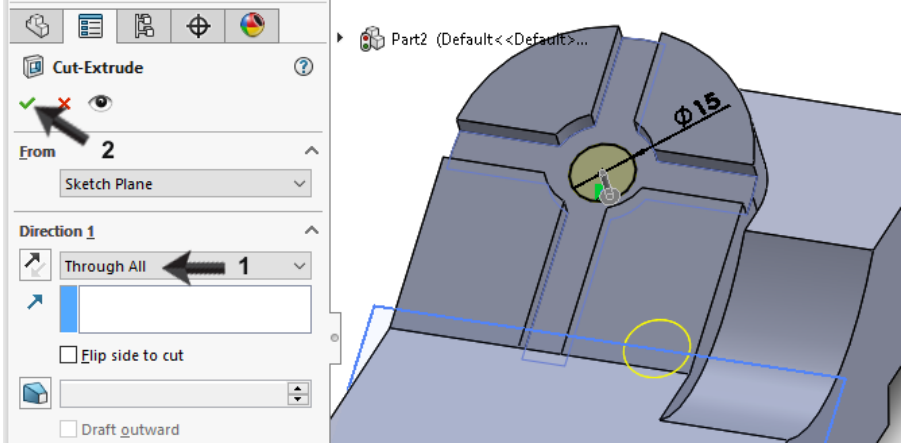
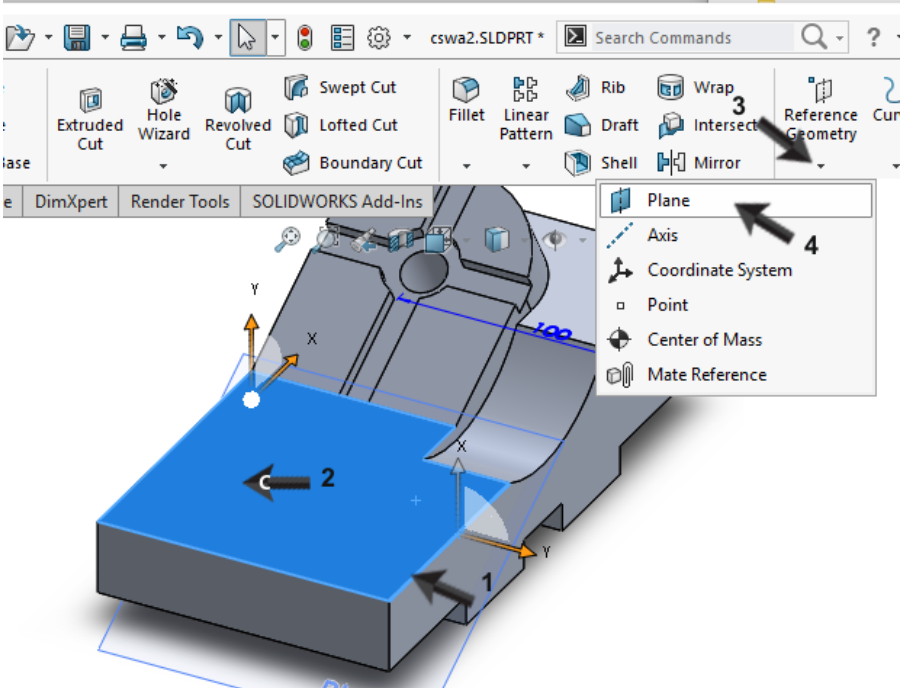
<p><b>58</b></p> <p>Make an extrusion from this sketch.</p> <ol style="list-style-type: none"> <li>1. Click on Reverse Direction in the Property-Manager to make sure that the extrusion goes downwards and not upwards.</li> <li>2. Select Up to Next to set the depth.</li> <li>3. Click OK.</li> </ol>	
<p><b>59</b></p> <p>Make the sketch as shown in the illustration on the right and continue to step 63.</p> <p>If you can not make this sketch by yourself, then follow the next few steps.</p>	
<p><b>60</b></p> <ol style="list-style-type: none"> <li>1. Select the plane to make a sketch on.</li> <li>2. Click on Arc in the CommandManager.</li> <li>3. Click on 3 Point Arc in the PropertyManager</li> <li>4. Set the first arc point at the corner as shown</li> <li>5. Set the second point on the edge</li> <li>6. Set the third point at a random position.</li> </ol>	

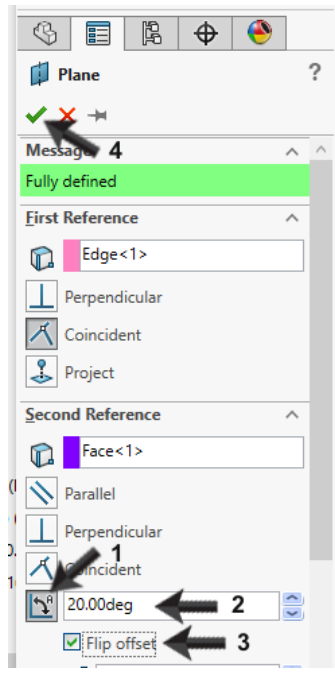
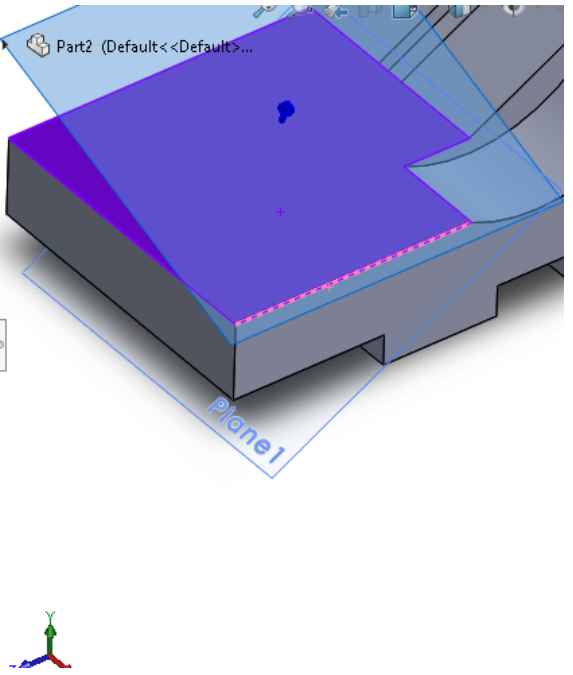
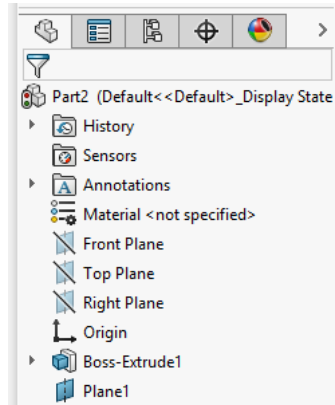
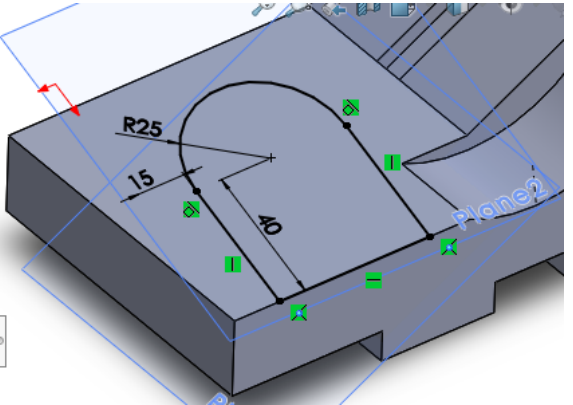
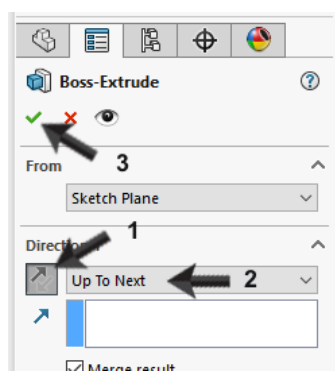
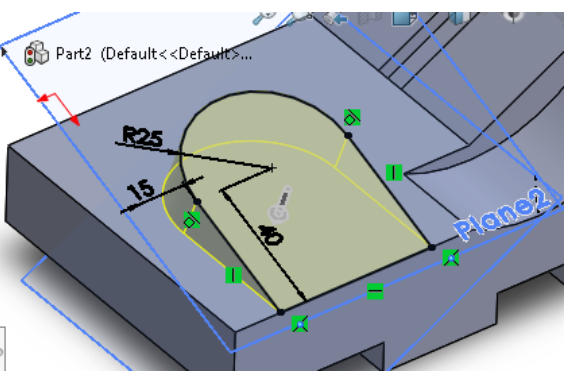
<p><b>61</b></p>	<p>Insert the two dimensions as shown.</p>	
<p><b>62</b></p>	<p>Draw two small lines above the arc as shown.</p>	
<p><b>63</b></p>	<p>Make an Extruded Cut from this sketch.</p> <ol style="list-style-type: none"> <li>1. Select the option Up To Surface to set the depth</li> <li>2. Click on the plane which indicates the end of the Cut Extrude.</li> </ol>	

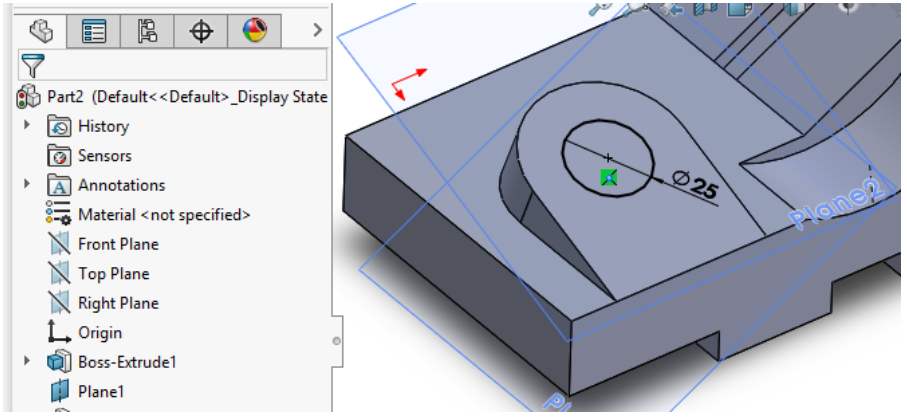
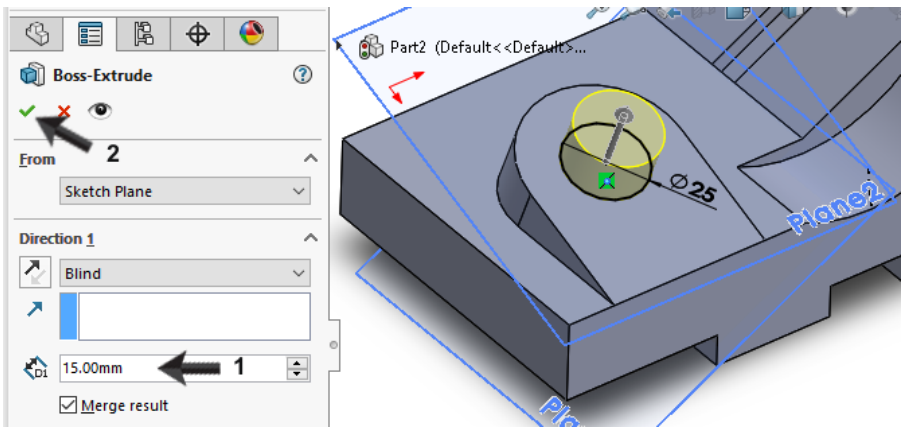
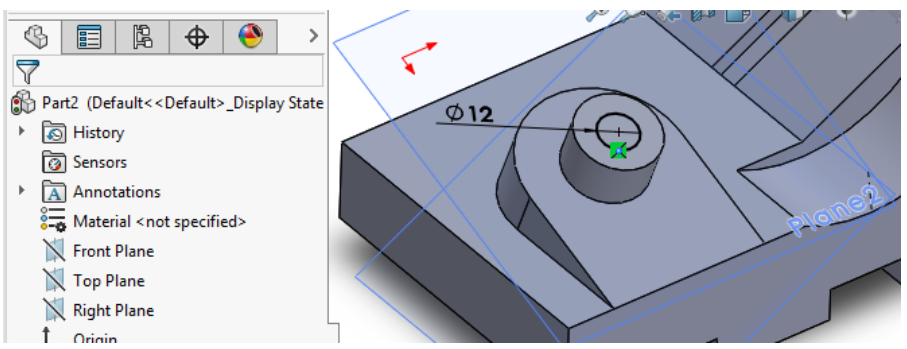
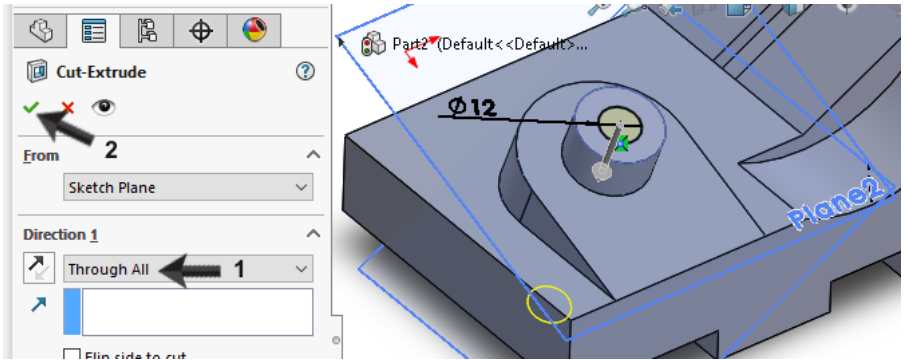


<p><b>67</b></p> <p>Draw two centerlines as shown on the right.</p> <p>Push &lt;esc&gt; after you have drawn the first centerline, next you have to draw the second centerline.</p>	
<p><b>68</b></p> <p>1,2 Select both centerlines (use the &lt;ctrl&gt;-key)</p> <p>3. Click on Offset in the CommandManager</p> <p>4. The distance is 5mm</p> <p>5. Select the option Bi-directional.</p> <p>6. Select the option Cap ends.</p> <p>7. Select the option Lines.</p> <p>8. Click OK.</p>	

<p><b>69</b></p> <ol style="list-style-type: none"> <li>1. Click on Trim Entities in the CommandManager.</li> <li>2. Click on the option Trim away inside in the PropertyManager.</li> <li>3. Click on the circle</li> <li>4. Click on all four lines that run through the circle. The pieces at the inside of the circle will be removed.</li> </ol>	
<p><b>70</b></p> <ol style="list-style-type: none"> <li>1. Click on Trim to Closest in the Property-Manager</li> <li>2. Click on the parts of the circle that you want to be removed.</li> </ol>	
<p><b>71</b></p> <p>Did you trim everything, then you can make an Extruded Cut from the sketch. Set the depth to 5mm.</p>	

<p><b>72</b></p>	<p>Make the sketch as shown in the illustration on the right.</p>	
<p><b>73</b></p>	<p>Make an Extruded Cut Through All.</p>	
<p><b>74</b></p>	<p>We will now make the second construction plane.</p> <ol style="list-style-type: none"> <li>1. Select the edge as shown.</li> <li>2. Hold the &lt;ctrl&gt;-key and select the plane as shown in the illustration.</li> <li>3. Click on the arrow beneath Reference Geometry in the CommandManager.</li> <li>4. Click on Plane.</li> </ol>	

<p><b>75</b></p> <ol style="list-style-type: none"> <li>1. In the PropertyManager, select the option Angle</li> <li>2. Set the angle of the new plane to 20°</li> <li>3. Click on Flip offset, so the plane extends in the right direction.</li> <li>4. Click OK.</li> </ol>		
<p><b>76</b></p> <p>Make a sketch as shown on the plane that you have just created.</p> <p>In steps 54 to 56 you have already made a similar sketch. If you want, you can check these steps to see how it is done.</p>		
<p><b>77</b></p> <p>Make an extrusion from the sketch.</p> <ol style="list-style-type: none"> <li>1. First click on Reverse Direction in the PropertyManager to extend the extrusion downwards.</li> <li>2. Select the option Up To Next.</li> <li>3. Click OK.</li> </ol>		

<p><b>78</b></p>	<p>Make the sketch as shown on the right.</p>	
<p><b>79</b></p>	<p>Extrude the sketch with a height of 15mm.</p>	
<p><b>80</b></p>	<p>Make the sketch as shown on the right.</p>	
<p><b>81</b></p>	<p>Make a Cut Extrude Through All from this sketch.</p>	

**82** The model is finished now. We will select the kind of material now.

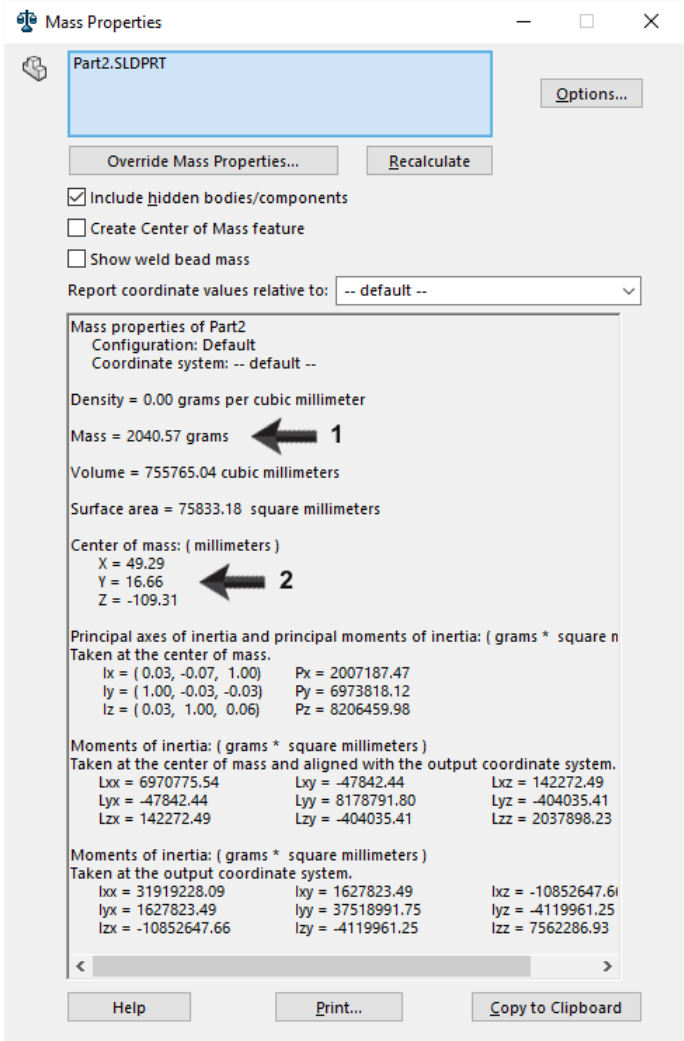
1. Right-click on Material in the FeatureManager.
2. Click on Edit Material.

---

**83**

1. Open the list of Aluminum Alloys in the PropertyManager.
2. Select 6061 Alloy
3. Verify if the density is the same as the one in the assignment.
4. Click OK.

Property	Value	Units
Elastic Modulus	69000	N/mm <sup>2</sup>
Poisson's Ratio	0.33	N/A
Shear Modulus	26000	N/mm <sup>2</sup>
Mass Density	2700	kg/m <sup>3</sup>
Tensile Strength	124.084	N/mm <sup>2</sup>
Compressive Strength		N/mm <sup>2</sup>
Yield Strength	55.1485	N/mm <sup>2</sup>
Thermal Expansion Coefficient	2.4e-005	/K
Thermal Conductivity	170	W/(m·K)

<p><b>84</b></p> <p>Finally we want to know the total mass from this part.</p> <p>Click on the tab Evaluate in the CommandManager and next on Mass Properties.</p> <p>In the popup-menu you can read a weight of 2040.57 grams. So this is answer <b>A</b> from the assignment.</p> <p>You can also see the Center of mass. This value is displayed in an X-, Y- and Z-coordinate in relation to the origin. In the model itself the center of mass is indicated too.</p>	 <p>Mass Properties dialog box for Part2.SLDPRT. The mass is 2040.57 grams. The center of mass coordinates are X = 49.29, Y = 16.66, and Z = -109.31 millimeters.</p>
<p><b>What are the main features you have learned in this tutorial?</b></p>	<p>As we have said in the introduction of this tutorial you did not learn a lot of new features. You have seen a few smart gadgets, though:</p> <ul style="list-style-type: none"> <li>• Some of the option from the offset-command</li> <li>• Some of the options from the Trim-command</li> <li>• The automatic change between a line and an arc shaped line while creating a sketch.</li> </ul> <p>More important is that you have seen and have practiced with two examples of CSWA-exams. You have seen how to make a plan when modeling a more complex part.</p> <p>By practicing a lot with this kind of assignments you can gain the necessary routine for passing the CSWA-exam and hence receiving a CSWA-certificate.</p>

# SOLIDWORKS works in education

You cannot imagine the modern technical world of today without 3D CAD. Whether your profession is in the Mechanical-, Electrical-, and Industrial Design- or Automotive industry: 3D CAD is THE tool of the designer or engineer from today.

SOLIDWORKS is the most used 3D CAD design software. Thanks to the unique combination of features: easy-to-use, widely applicable and with an excellent support. In the annual updates more and more customer wishes are implemented, which leads to an annual increase of the functionality, but also to optimization of functions already available in the software.

## Education

A great number of educational institutes, in a variety from Technical Vocational Training to Universities already have chosen for SOLIDWORKS. Why?

For a **tutor** the choice for SOLIDWORKS is a choice for user-friendly software, easy to learn for pupils and students. SOLIDWORKS fits into the system of a problem-initiated training or a competence-related training. Tutorials are available for the different levels of training, like a series of tutorials for Technical Vocational level in which the scholar is lead through the software step-by-step. Also the higher levels, in which complex designing - for instance double curved planes - is needed, can work with SOLIDWORKS. All tutorials are in English and free-downloadable from [www.SOLIDWORKS.com](http://www.SOLIDWORKS.com).

For a **scholar** or a **student**, learning to work with SOLIDWORKS is fun and defying. By using SOLIDWORKS, technique becomes more and more visible and tangible, which results in a more fun and realistic way of working on an assignment. Even better, every scholar or student knows that job-opportunities increase when SOLIDWORKS, the most used 3D-CAD software is on his or her resume. On many job sites you will find a great number of available jobs and internships that require SOLIDWORKS. This will increase the 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 tutor.

The choice to work with SOLIDWORKS is an important issue for the **ICT-department** because the

need to install new hardware can be postponed thanks to the fact that SOLIDWORKS has relatively low hardware demands. The installation and management of SOLIDWORKS in a network is very simple, amongst others because of the use of network licenses. And if a problem occurs after all, a qualified helpdesk is available, which will help you to get back on the right track again.

## Certification

When you control SOLIDWORKS sufficiently you can join the CSWA-test. CSWA stands for Certified SOLIDWORKS Associate. After passing this exam, you will receive a certificate which can be used to prove that you are in control of SOLIDWORKS. This can be very useful when applying for a job or internship.

After finishing this series of tutorials, you will know enough to join the CSWA-test.

## Finally

SOLIDWORKS has committed itself for an extended period to educational institutes and schools. By supporting teachers where possible, making tutorials available, adapting the software annually to the latest version and by supplying the Student Kit. The choice for SOLIDWORKS is a choice for the future. The future of education, which ensures itself of a wide support and a future of scholars and students, who want to have the best opportunities after their technical training.

## Contact

Do you still have questions about SOLIDWORKS, please contact your local reseller.

Please visit our website for more information on SOLIDWORKS: <http://www.SOLIDWORKS.com>