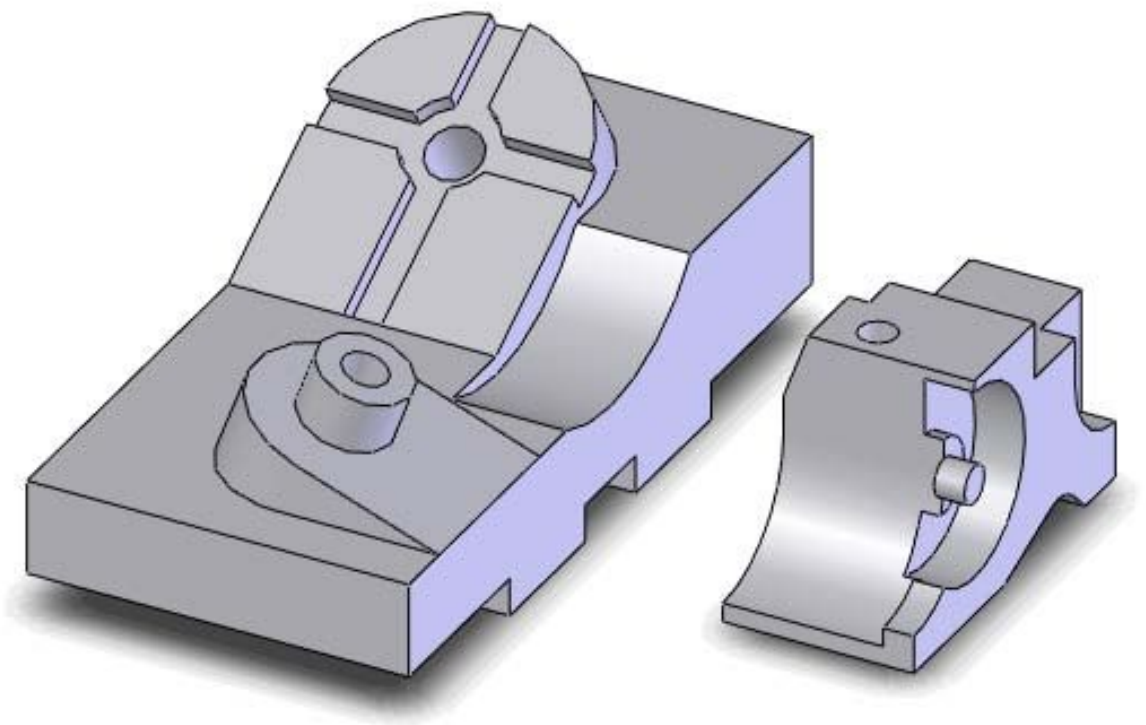
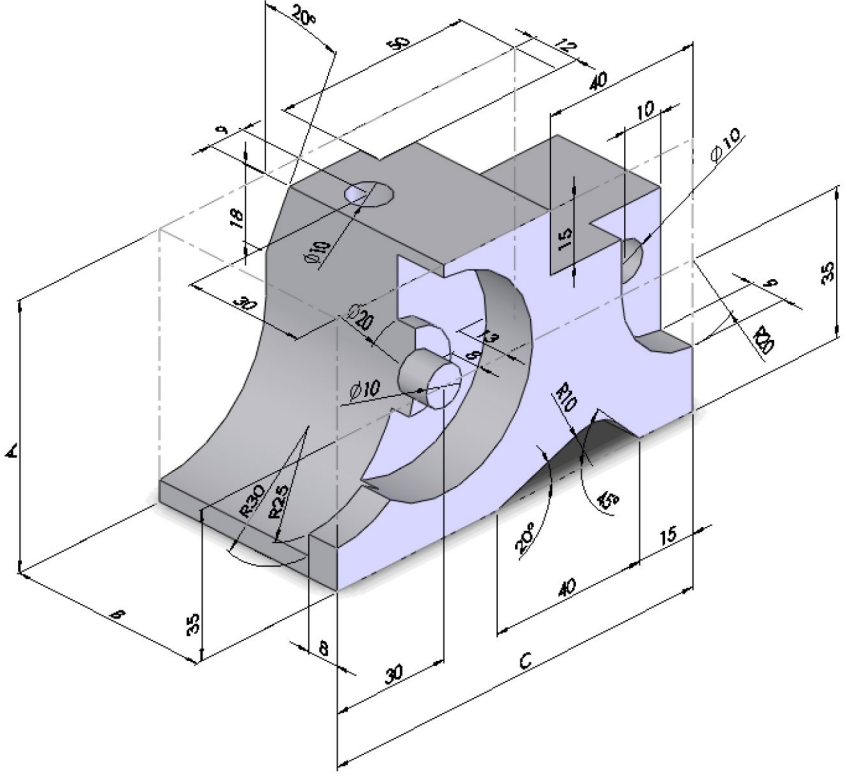


Tutorial 1-2 Design



Tutorial 1

	We will show you how to complete an assignment as described on the test.	
	Build this part in SolidWorks.	Your assignment is to build a part in SolidWorks.
	Unit system: MMGS (millimeter, gram, second).	Although inches are often used in the United States, we will work in millimeters and grams, using the metric system, which is the default unit system 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.

	<p>All holes through all, unless otherwise specified.</p>	<p>All holes will go through the whole model, unless otherwise specified (this is often not visible in the drawing or illustration).</p>
	<p>Part material: Copper. Density = 0.0089 g/mm³.</p>	<p>The part is made out of copper in this example. The specific weight of copper is 0.0089gram per mm³.</p>
	<p>What is the overall mass of the part in grams?</p> <ul style="list-style-type: none"> a. 1205 b. 1280 c. 144 d. 1108 	<p>What is the total weight of the part? It is a multiple choice question with four possible answers.</p>
		
<p>Work plan</p>	<p>Although the shape of this assignment looks fairly complicated at first glance, you will see that it is built using boss-extrude and cut-extrude commands. The hardest part of the assignment is making a work plan for it. Look at the shape very closely and try to divide it into different features. It is very important to do this before you start modeling! Below you will learn the steps we have used to build the model. Every step is a feature that we will make. There are 10 features In total.</p>	

<p>1</p>	<p>Start SolidWorks and open a new part.</p>	
<p>2</p>	<p>Select the Right Plane and make a sketch as shown on the right.</p> <p>Can you make this sketch yourself already? Very good. Continue with Step 8.</p> <p>If you do not succeed doing it yourself, try using the next few steps.</p>	
<p>3</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 50mm. With this as a base the proportions will be right.</p>	

<p>4 Draw the arc now:</p> <ol style="list-style-type: none"> 1. Click on Arc in the CommandManager. 2. Click on Tangent Arc in the PropertyManager. 3. Click on the lower end of the vertical line as shown on the right. 4. Click on the upper end of the vertical line as shown on the right. 	
<p>5 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"> 1. Select the midpoint of the arc. 2. Select (holding the <Ctrl> key) the left vertical line. 3. Click on 'Coincident' in the CommandManager. 	

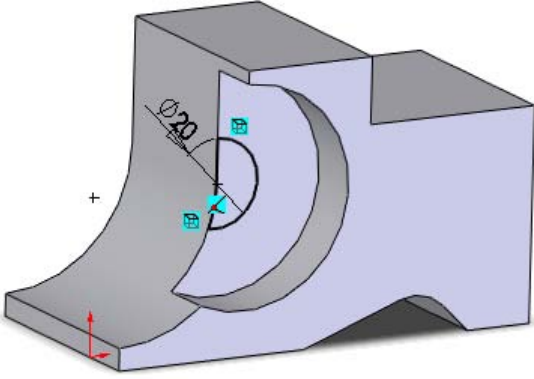
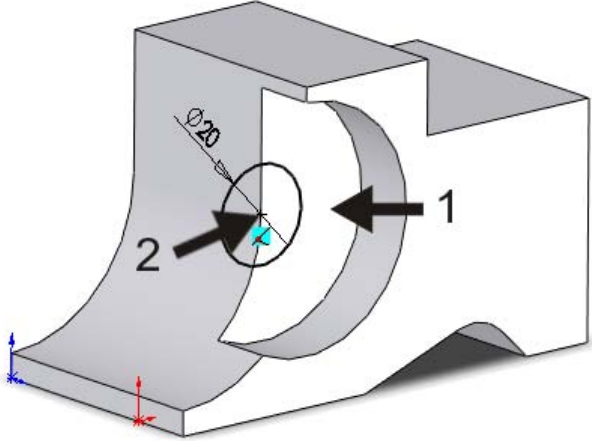
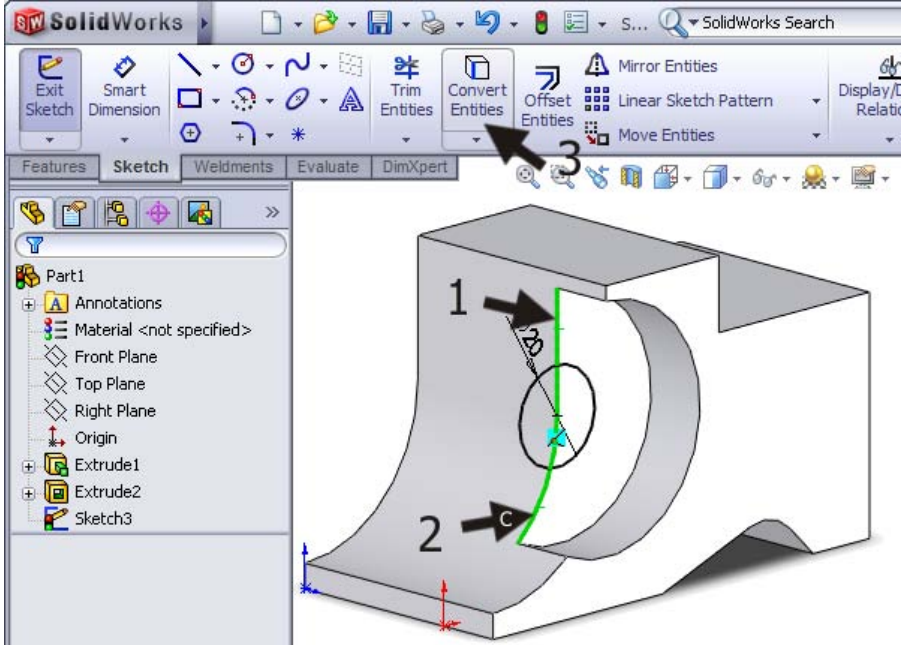
<p>6 Make a fillet at the bottom of the sketch:</p> <ol style="list-style-type: none"> Click on Sketch Fillet in the CommandManager. Check to make sure you have set a radius of '10mm' in the PropertyManager (this is the default value). Click on the corner you want to fillet in the sketch. 	
---	--

<p>7 Set the dimensions in the sketch as shown on the right.</p>	
---	--

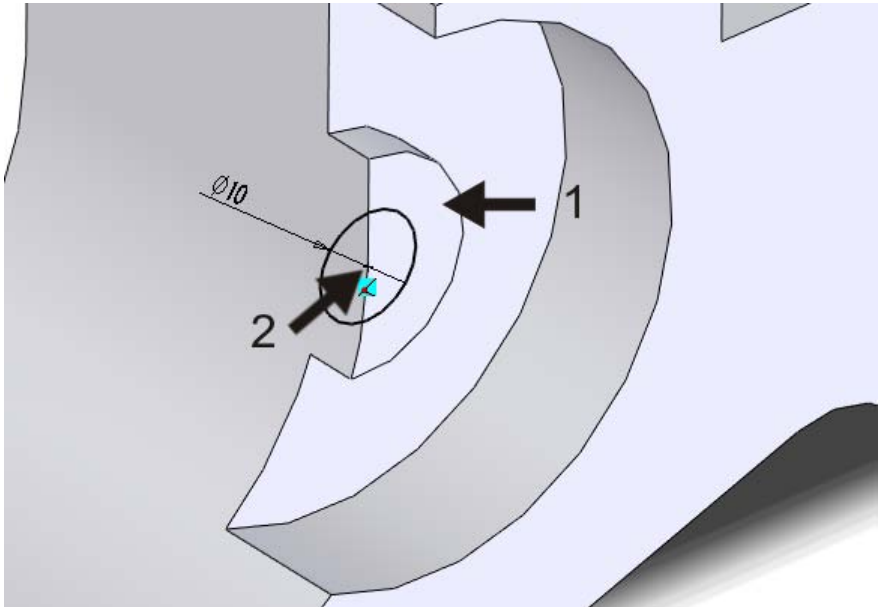
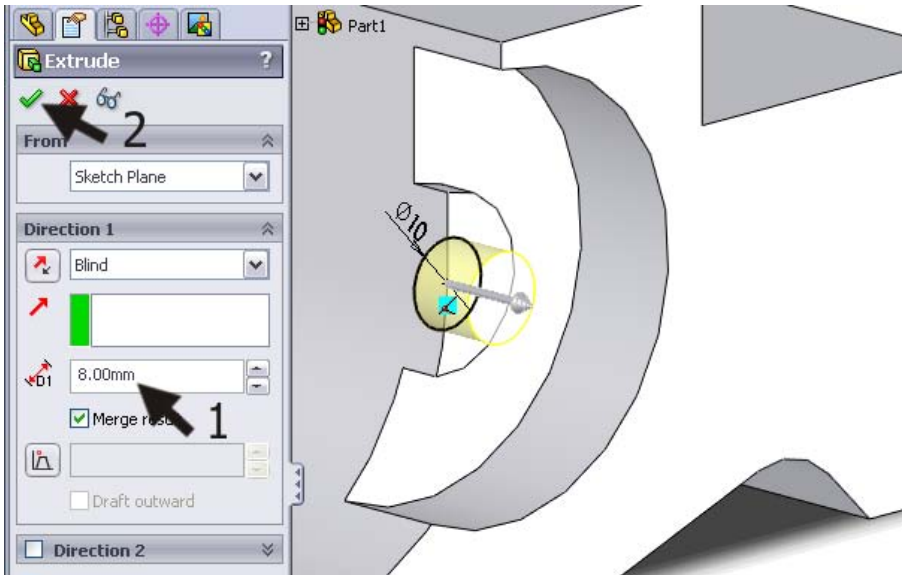
<p>8</p>	<p>Extrude the sketch to '50mm'.</p>	
-----------------	--------------------------------------	--

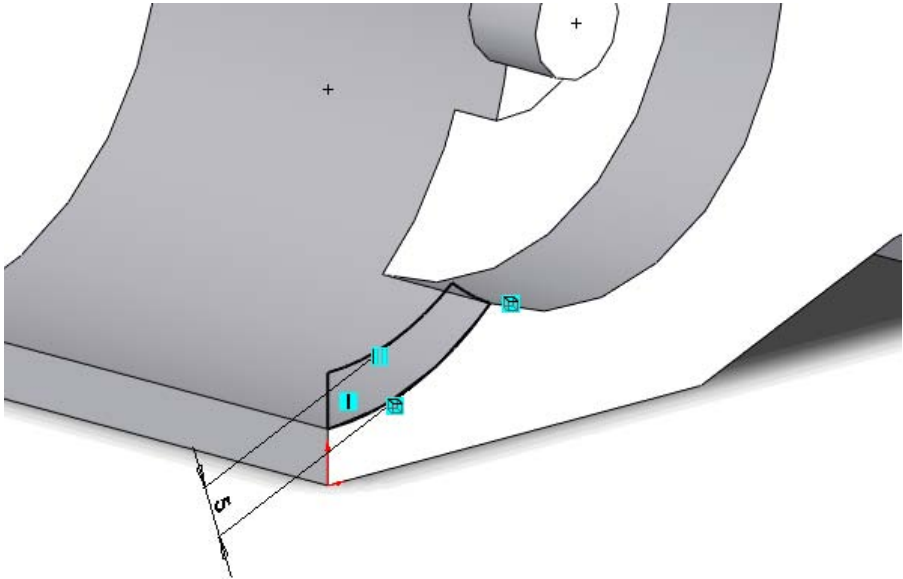
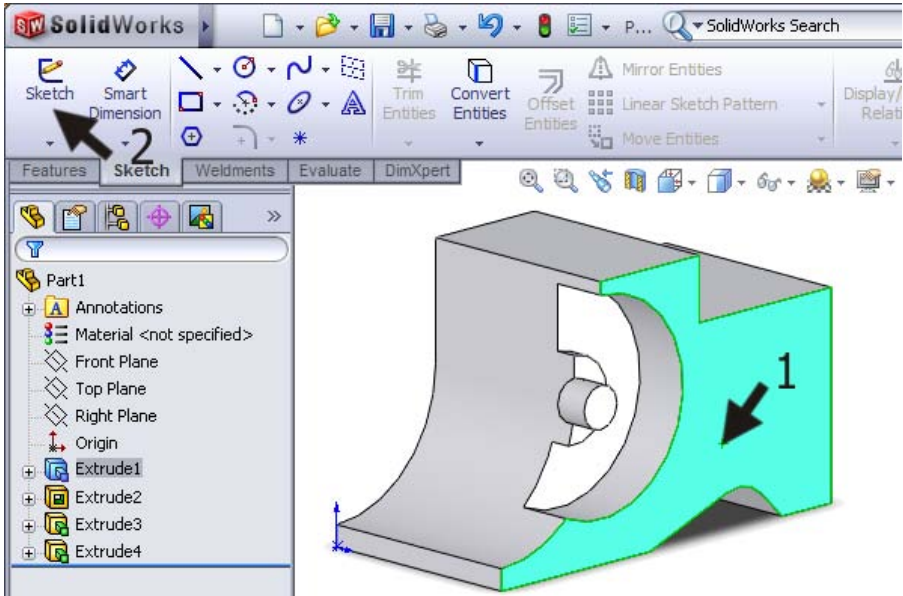
<p>9</p> <p>Next, make a sketch, as shown on the right.</p> <ol style="list-style-type: none"> 1. Select the front surface of the model to draw a new sketch on it. 2. Click on the point where the line converts into the arc. <p>Draw the circle and set the dimension in the sketch.</p>	
--	--

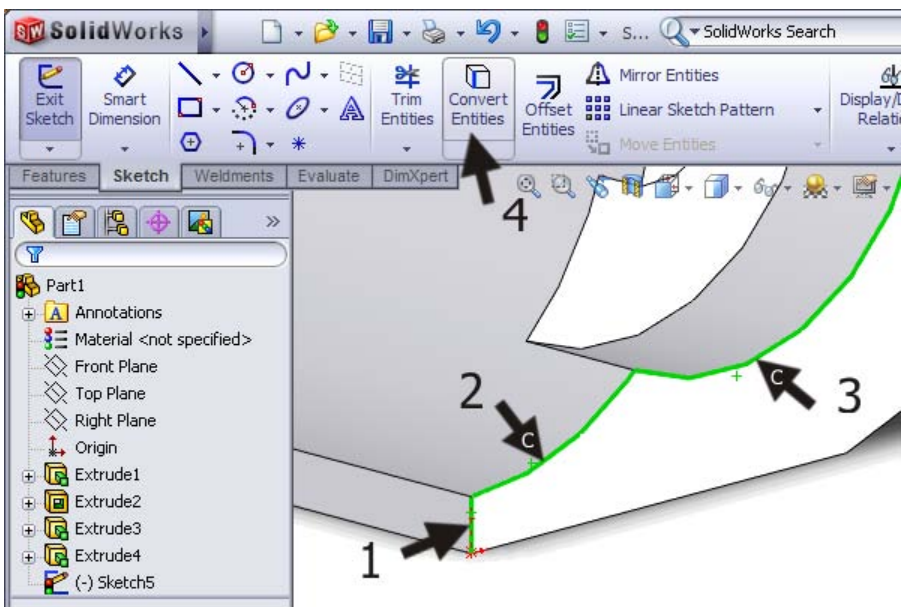
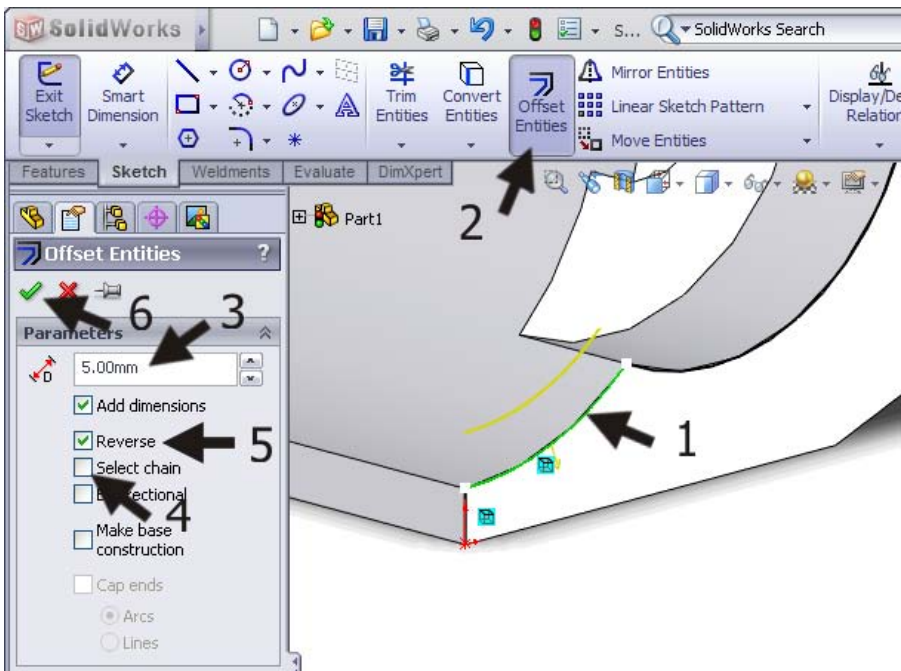
<p>10</p>	<p>Make an Extruded Cut from the sketch, setting the depth to '13mm'.</p>	
------------------	--	--

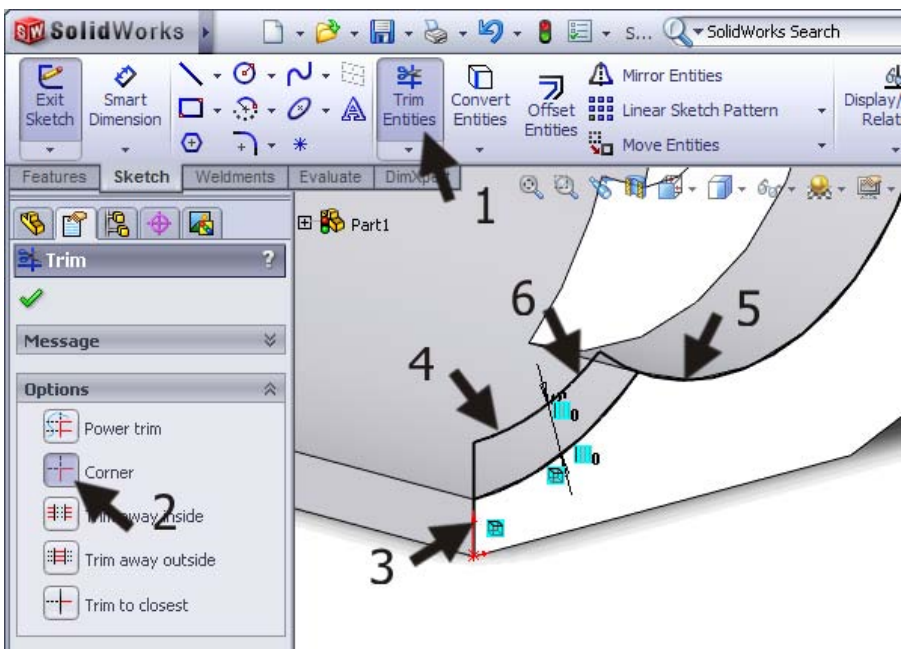
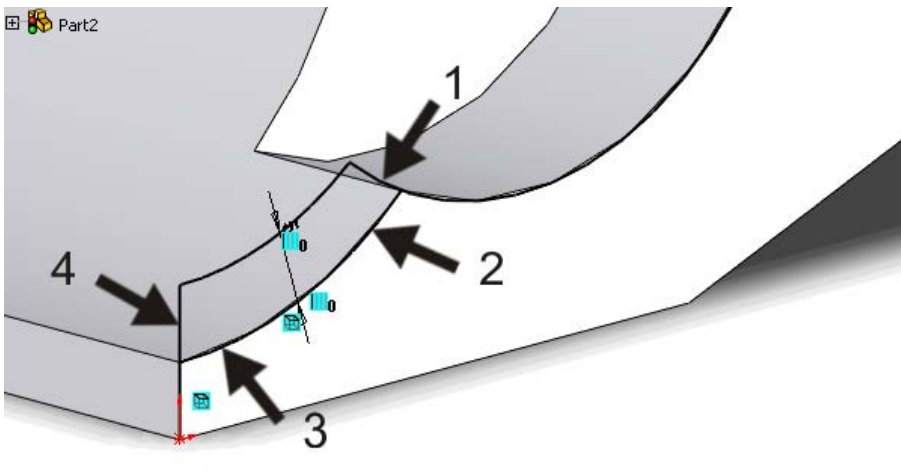
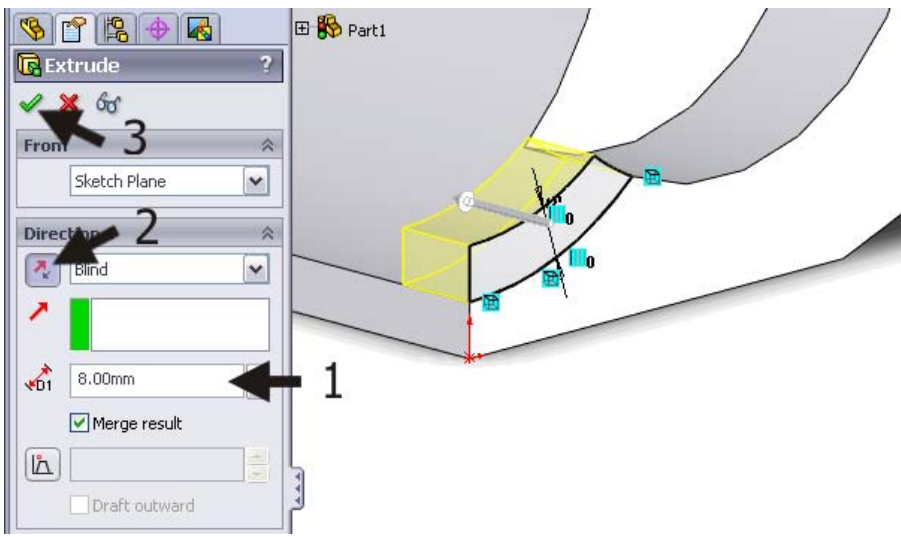
<p>11 Make a sketch as shown on the right.</p> <p>Can you do it yourself? Proceed to Step 15.</p> <p>If this does not work out, watch the following steps, which tell you how to handle this.</p>	
<p>12</p> <ol style="list-style-type: none"> 1. Select the deeper plane first. On this surface we will make a new sketch. 2. Draw a circle and make sure the midpoint is exactly at the point where the straight line converts in to an arc. 3. Set the size of the circle to 'Ø20mm'. 	
<p>13</p> <p>Push the <Esc> key on your keyboard to end the 'Smart Dimension' command.</p> <ol style="list-style-type: none"> 1,2 Select the line and the arc as shown on the right. 3. Click on 'Convert Entities' in the Command-Manager. 	

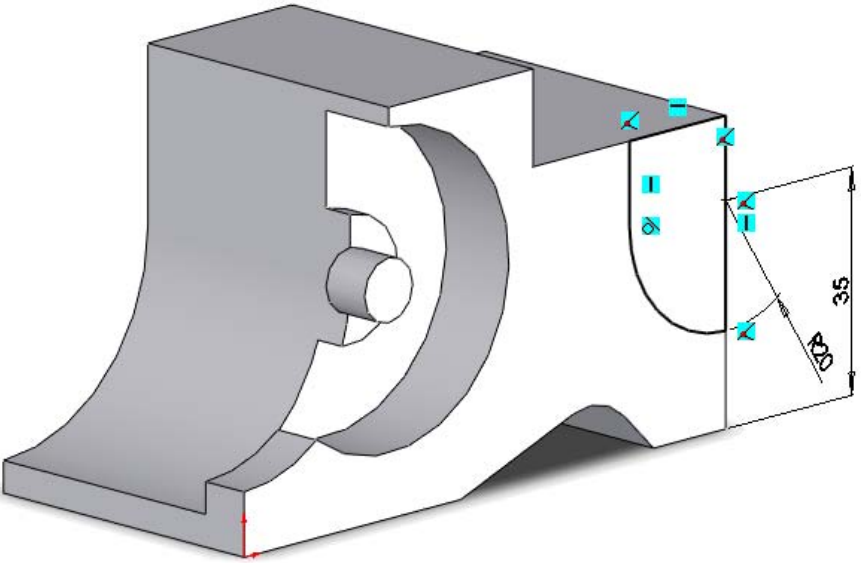
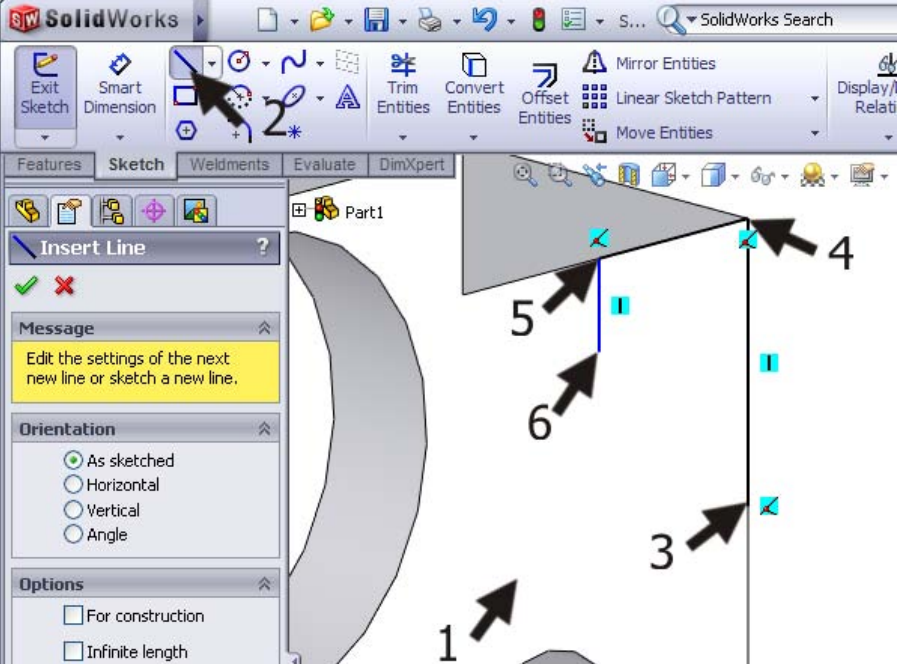
<p>14</p>	<ol style="list-style-type: none"> 1. Click on 'Trim Entities' in the CommandManager. 2. Click on 'Trim to closest' in the PropertyManager. 3. Click on the three parts of the sketch that need to be removed. 	
<p>15</p>	<p>Extrude this sketch to a depth of '5mm'.</p>	

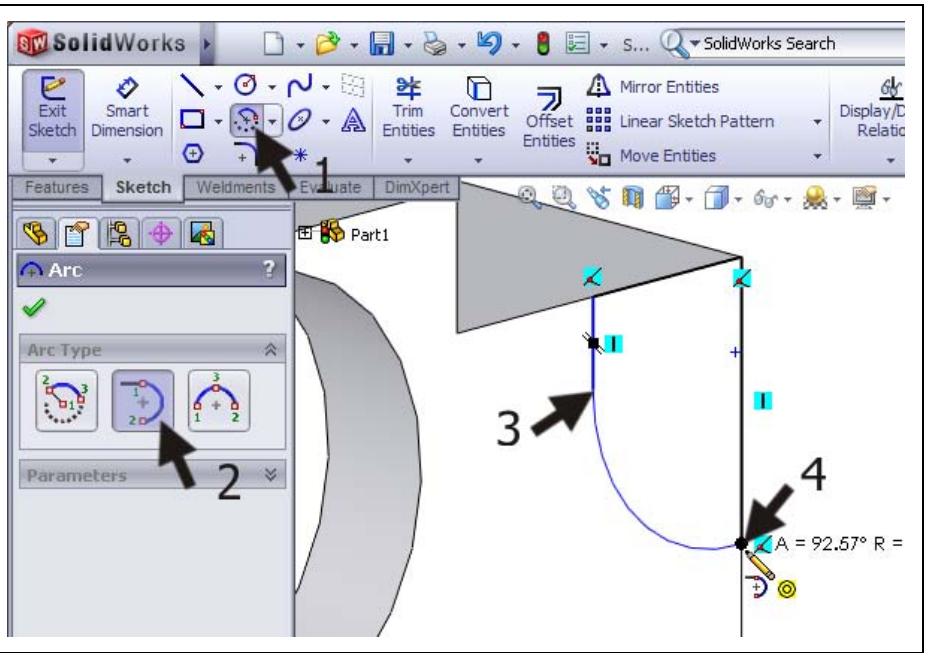
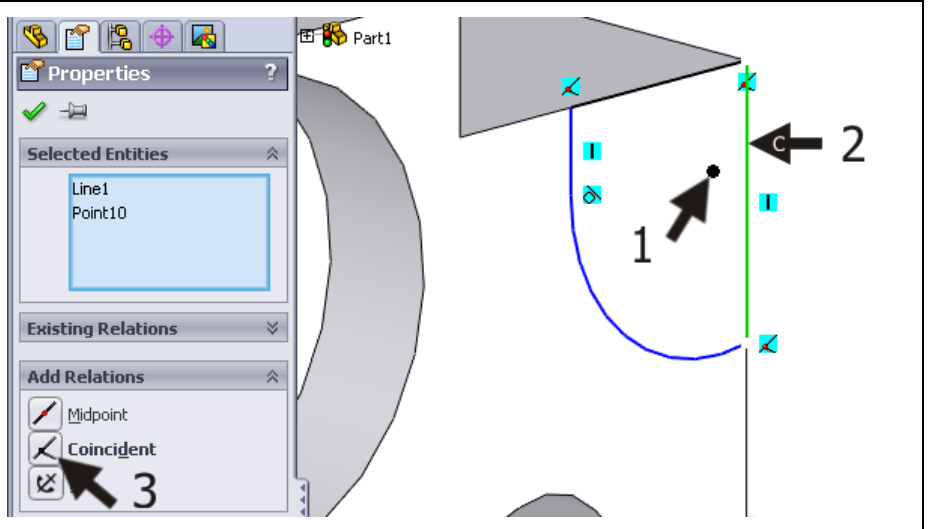
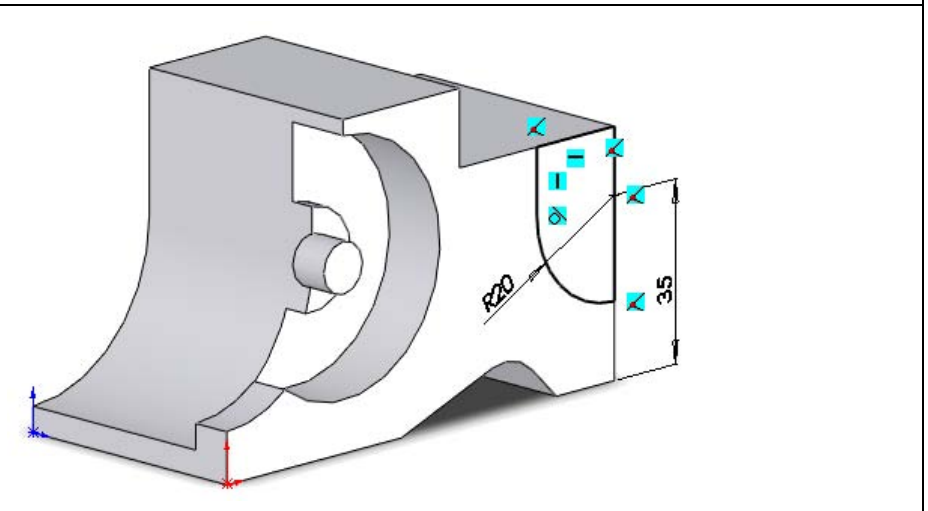
<p>16 Make the sketch as in the illustration on the right.</p> <ol style="list-style-type: none"> 1. Select the plane to draw a sketch on. 2. Draw a circle. Make sure the midpoint is exactly on the point where the straight line converts into an arc. 3. Set the size of the circle to '$\varnothing 10\text{mm}$'. 	
<p>17 Extrude the sketch to a depth of '8mm'.</p>	

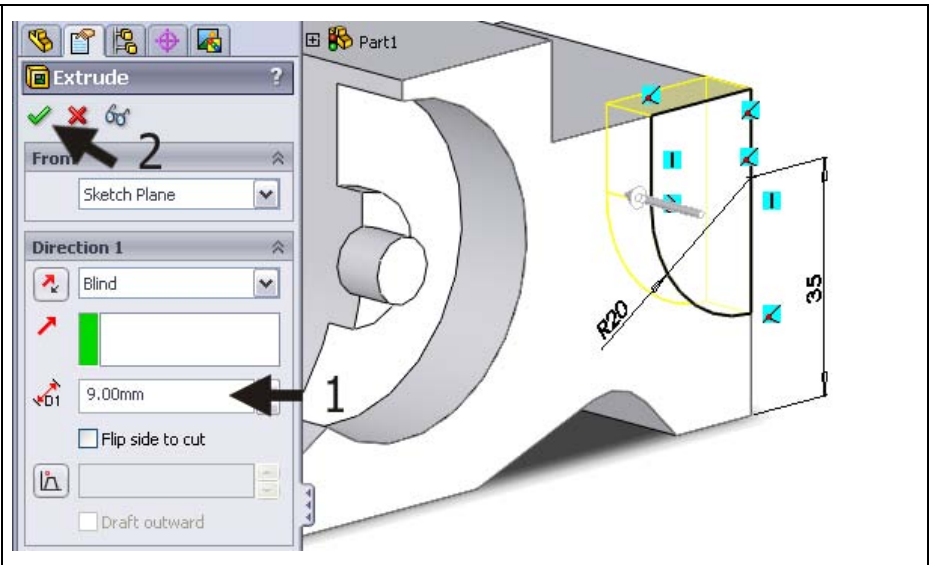
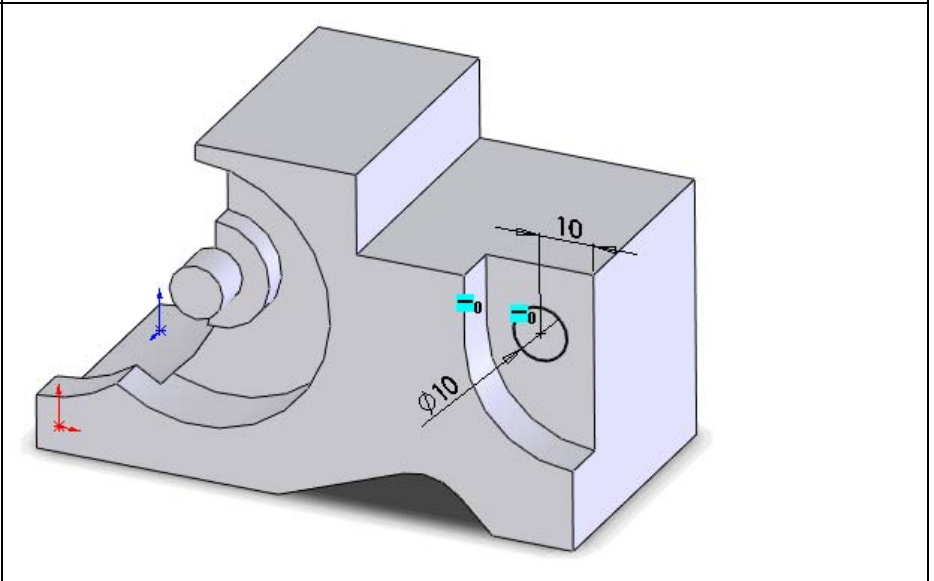
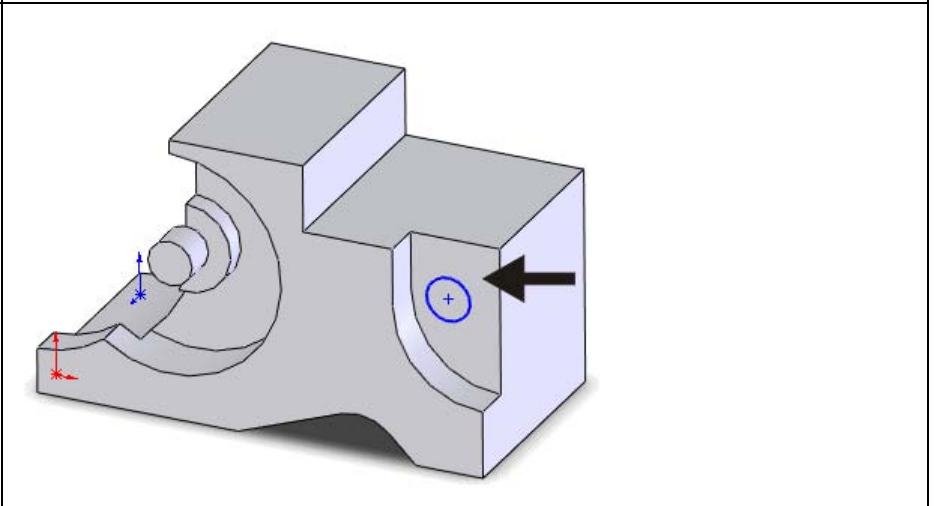
<p>18</p> <p>Make the sketch as drawn on the right.</p> <p>Can you manage it yourself? If you can, proceed to Step 24.</p> <p>If you cannot do it all by yourself, follow the next steps.</p>	
<p>19</p> <ol style="list-style-type: none"> 1. Select the plane you want to make a sketch on. 2. Click on 'Sketch' in the CommandManager to open the sketch. 	
<p>Tip!</p>	<p>In most cases 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 opened the sketch explicitly. Why? Because we will use the Convert Entities command first and the sketch must be open to use this command. That is the reason for this action.</p>

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

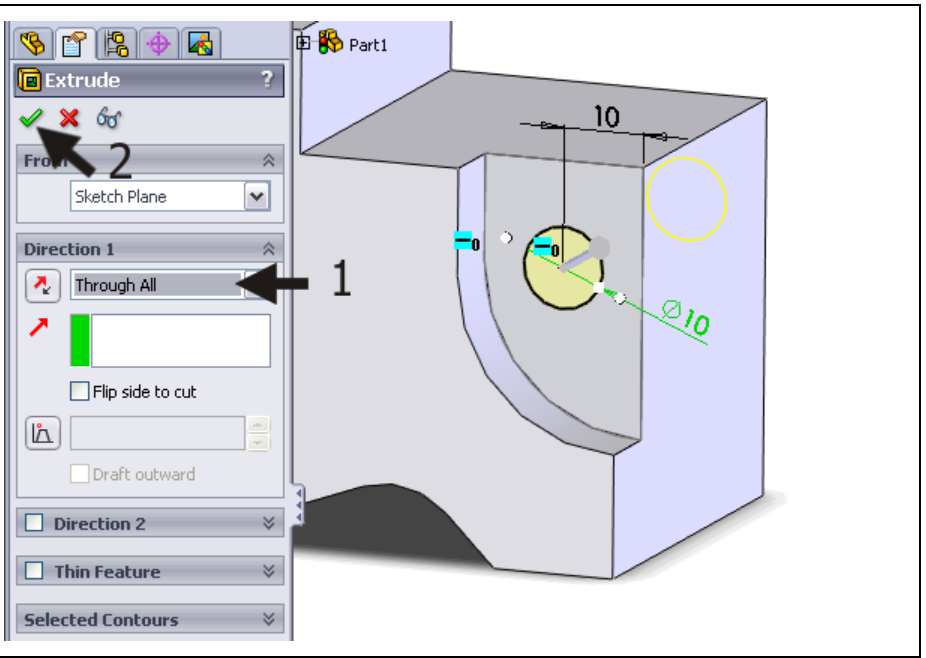
<p>22</p> <ol style="list-style-type: none"> 1. Click on 'Trim Entities' in the CommandManager. 2. Select the option 'Corner' in the PropertyManager. 3-6 Make the upper corners by clicking as indicated in the illustration. 	
<p>23</p> <p>Next, make the bottom corner points by clicking as shown on the right.</p>	
<p>24</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>	

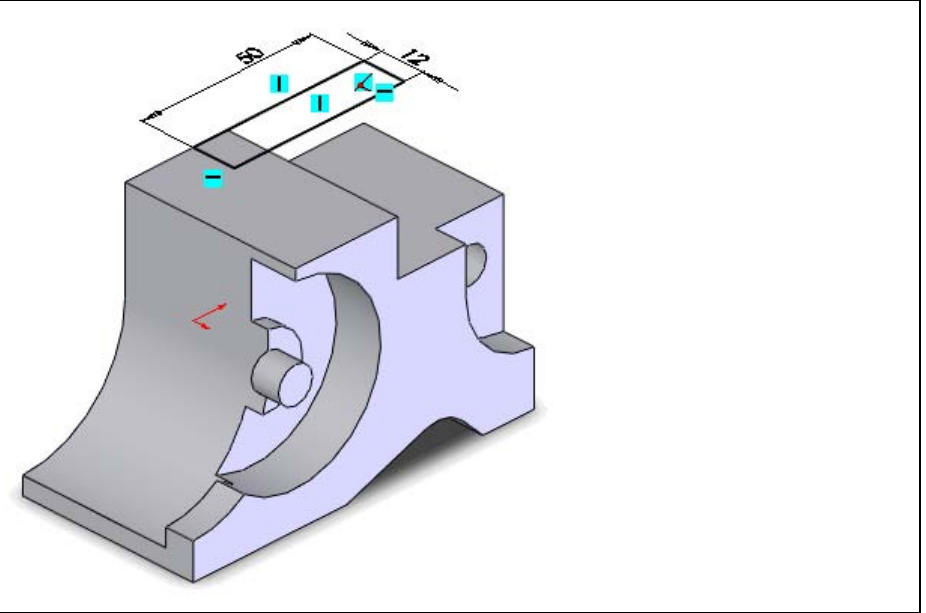
<p>25</p>	<p>Make the sketch as shown.</p> <p>Can you manage this by yourself? Continue to Step 30. If not, follow the next few steps.</p>	
<p>26</p>	<p>Select the plane on which you want to make a sketch.</p> <p>Draw three straight lines as shown in the illustration.</p>	

<p>27</p> <ol style="list-style-type: none"> 1. Click on Tangent Arc in the CommandManager. 2. Click at the bottom end of the left vertical line. 3. Click on the bottom end of the right vertical line. 	
<p>28</p> <ol style="list-style-type: none"> 1. Select the midpoint of the arc. 2. Hold the <Ctrl>-key and select the right vertical line too. 3. Click on 'Coincident' in the PropertyManager. 	
<p>29</p> <p>Add the two dimensions as shown.</p>	

<p>30</p>	<p>Make an Extruded Cut from this sketch with a depth of '9mm'.</p>	
<p>31</p>	<p>Make the sketch as shown and continue to Step 35. If you cannot manage this yourself, follow the next few steps.</p>	
<p>32</p>	<ol style="list-style-type: none"> 1. Select the plane to make the next sketch as shown on the right. 2. Draw a circle, just about the size and position as in the illustration. 	

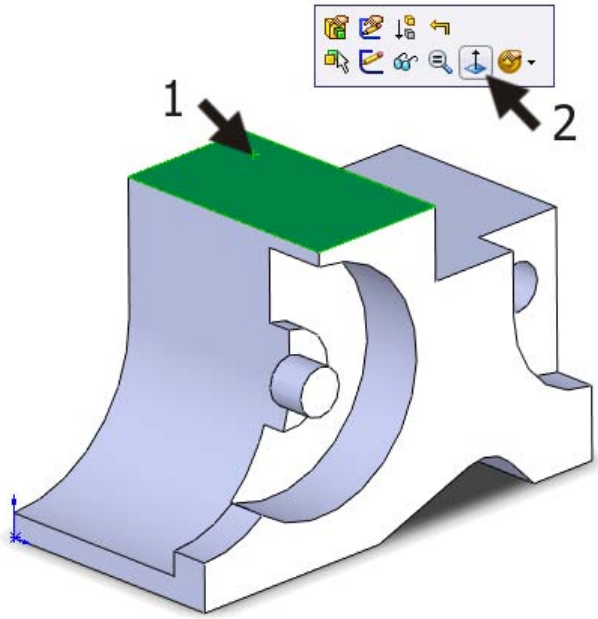
<p>33</p>	<ol style="list-style-type: none"> 1. Select the midpoint from the circle. 2. Hold the <Ctrl>-key and click on the point as shown on the right. 3. Click on 'Horizontal' in the PropertyManager. 	
<p>34</p>	<p>Set the sizes as shown in the illustration.</p>	

<p>35 Make an Extruded Cut from this sketch. Select the option 'Through All'.</p>	
--	--

<p>36 Make the sketch as shown on the right and continue to Step 40. If you cannot manage this yourself, follow the next few steps.</p>	
--	---

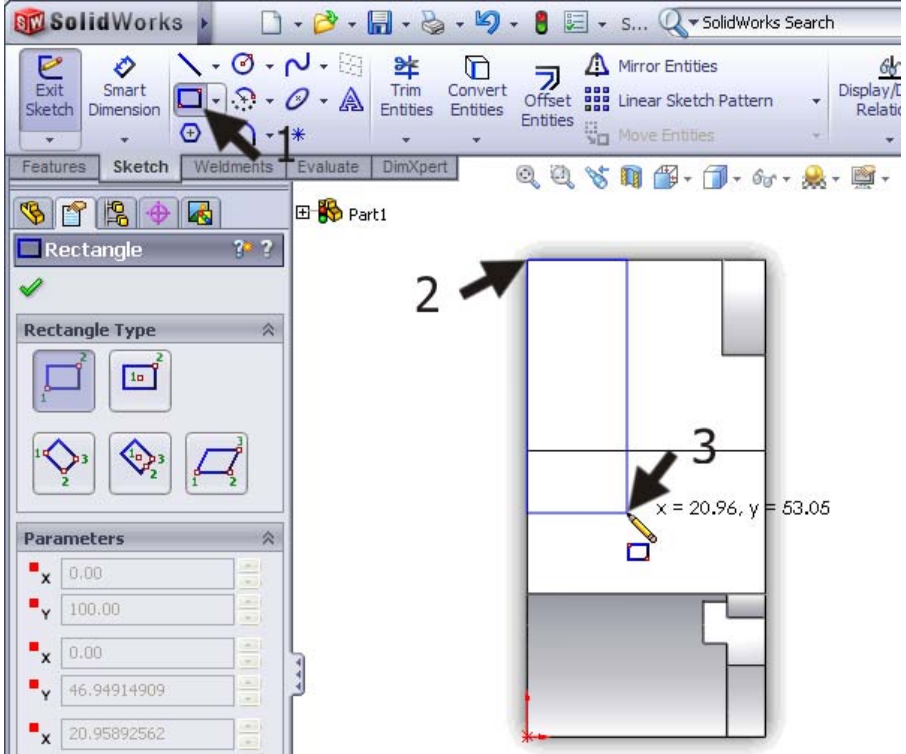
37

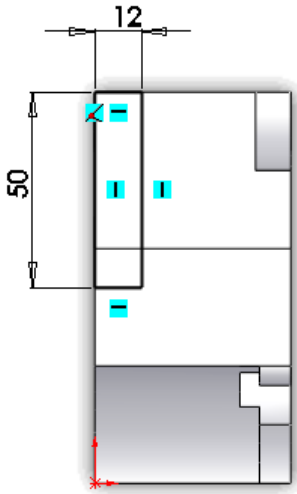
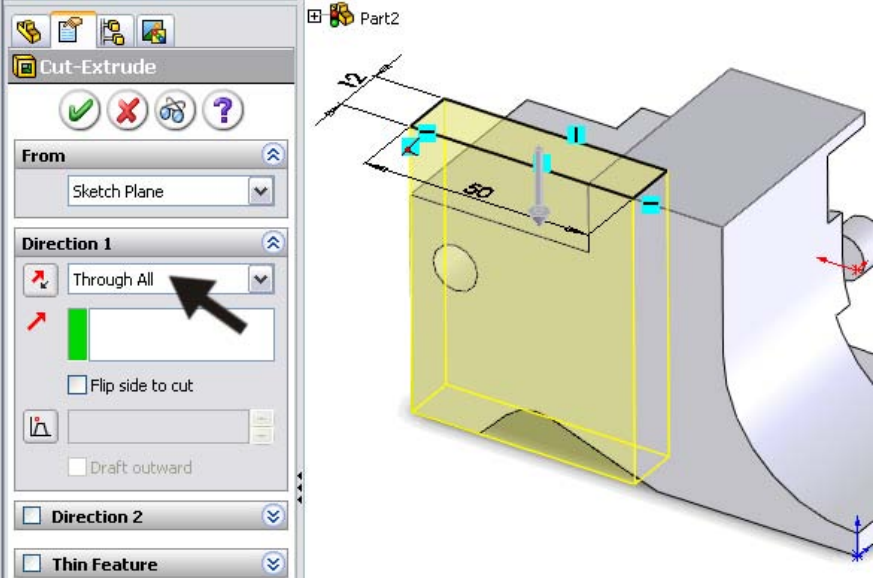
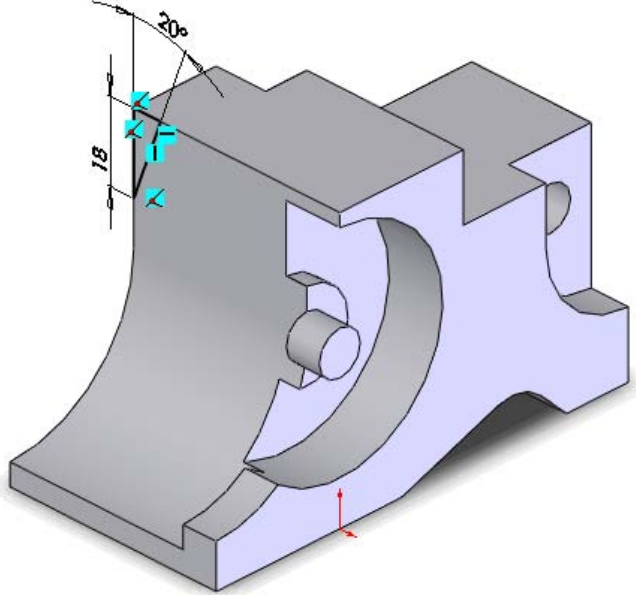
1. Select the upper surface from the model
2. Click on **Normal To** in the pop-up menu.

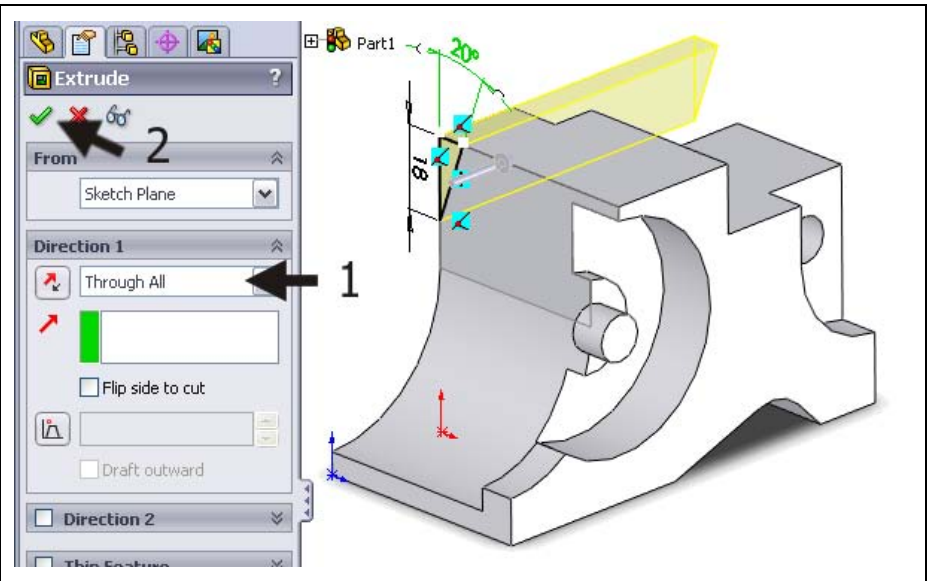
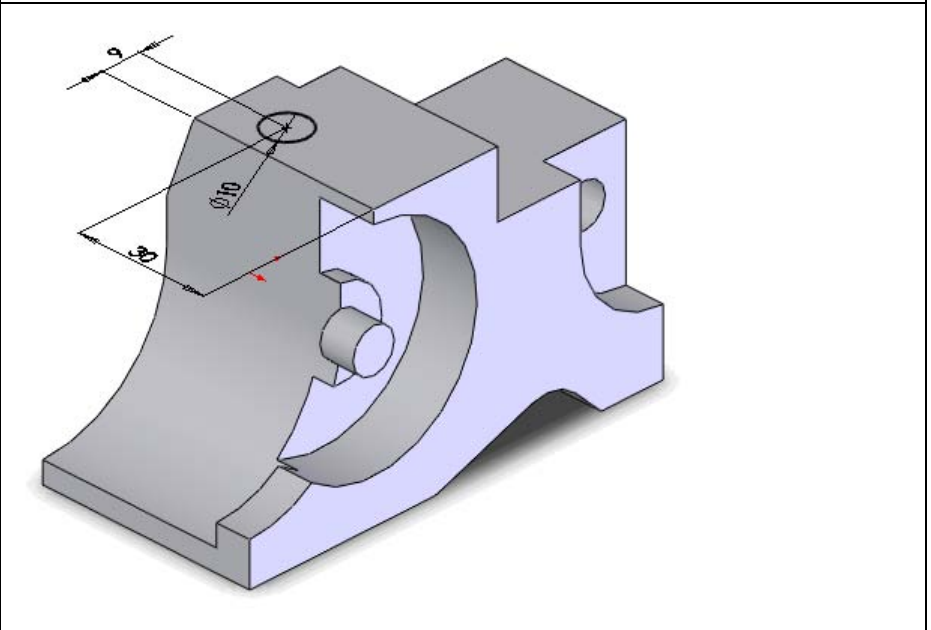
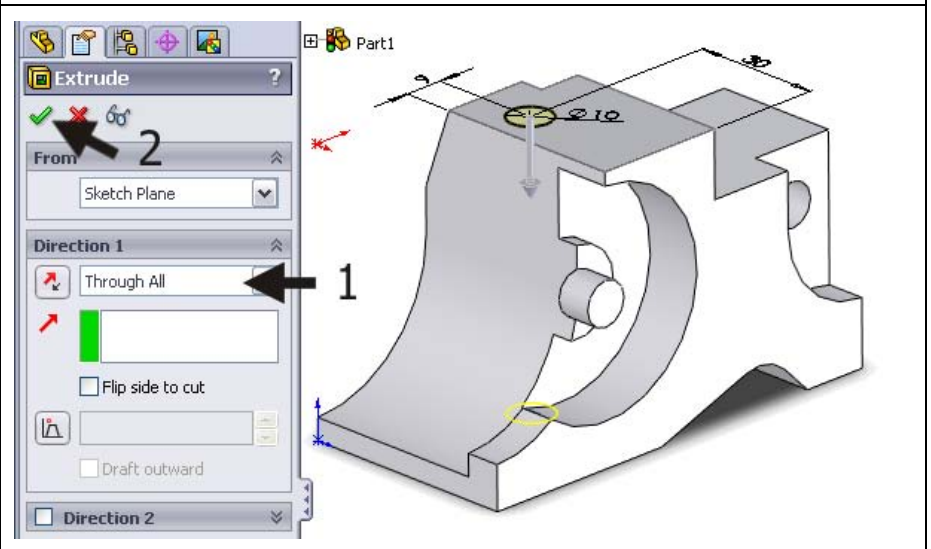


38

1. Click on **Rectangle** in the **CommandManager**.
2. Draw the rectangle as shown in the illustration on the right.



<p>39</p>	<p>Set the two sizes as shown.</p>	 <p>A 2D sketch of a rectangular feature on a grey base. The width of the rectangle is dimensioned as 12, and its height is dimensioned as 50. There are several cyan dimension lines and arrows indicating the placement of the rectangle. A red arrow at the bottom indicates the extrusion direction.</p>
<p>40</p>	<p>Make an Extruded Cut from this sketch and set the depth to 'Through All'.</p>	 <p>The image shows the 'Cut-Extrude' dialog box in a CAD software. The 'From' dropdown is set to 'Sketch Plane'. Under 'Direction 1', the 'Through All' option is selected, indicated by a black arrow. Other options like 'Flip side to cut', 'Draft outward', 'Direction 2', and 'Thin Feature' are visible. To the right, a 3D model of a grey part shows a yellow rectangular cut being applied to its top surface, with dimensions 12 and 50 visible on the cut.</p>
<p>41</p>	<p>Make the sketch as shown.</p>	 <p>A 3D model of a grey mechanical part. A chamfered edge is shown with a 30-degree angle dimension. A vertical dimension of 18 is shown on the left side of the part. Cyan dimension lines and arrows are used to indicate the chamfer and the 18mm dimension. A red arrow at the bottom indicates the extrusion direction.</p>

<p>42</p>	<p>Make an Extruded Cut from this sketch and set the depth to 'Through All'.</p>	
<p>43</p>	<p>Finally, make the sketch as shown in the illustration on the right.</p>	
<p>44</p>	<p>Make an Extruded Cut from this sketch and set the depth to 'Through All'.</p>	

<p>45 The model is now ready. We have to select the kind of material, and the assignment says 'copper'.</p> <ol style="list-style-type: none"> 1. Right-click on 'Material' in the FeatureManager. 2. When 'Copper' is in the list, you can click on it. If not, click on 'Edit Material'. 	
<ol style="list-style-type: none"> 1. Open the list 'Copper and its Alloys' in the PropertyManager. 2. Select 'Copper'. 3. Just to be sure: check the density under 'Physical Properties'. Is it the same as in the assignment? 4. Click on OK. 	

47 We want to know the weight of this part:

1. Click on the tab 'Evaluate' in the CommandManager.
2. Click on 'Mass Properties'.

48 In the pop-up menu you can read the weight: '1280.33 grams'.

Mass properties of Part1 (Part Configuration - Default)

Output coordinate System: -- default --

Density = 0.01 grams per cubic millimeter

Mass = 1280.33 grams

Volume = 143857.58 cubic millimeters

Surface area = 26112.48 millimeters²

Center of mass: (millimeters)

X = 26.81
Y = 25.80
Z = -56.06

Principal axes of inertia and principal moments of inertia: (grams * square millimeters)
Taken at the center of mass.
Ix = (-0.13, -0.14, 0.98) Px = 554808.12

Tutorial 2

We are going to build a second model. Again, this is an assignment similar to the first one.

Build this part in SolidWorks.

Unit system: MMGS (millimeter, gram, second).

Decimal places: 2.

Part origin: Arbitrary.

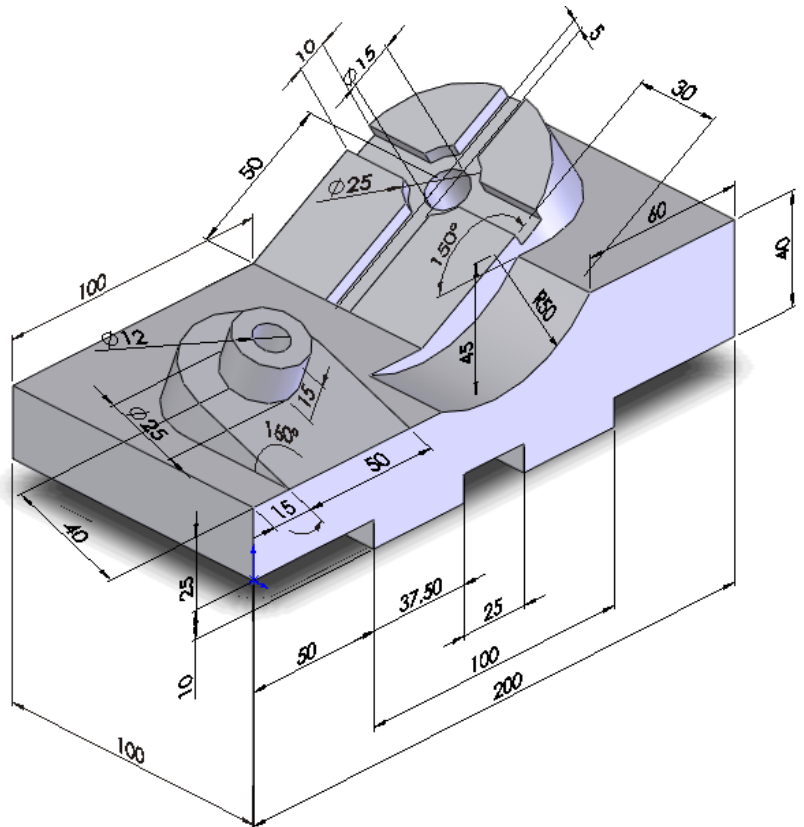
All holes through all, unless otherwise specified.

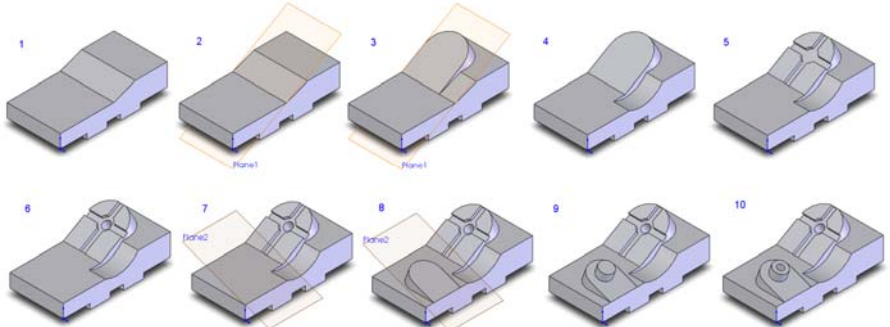
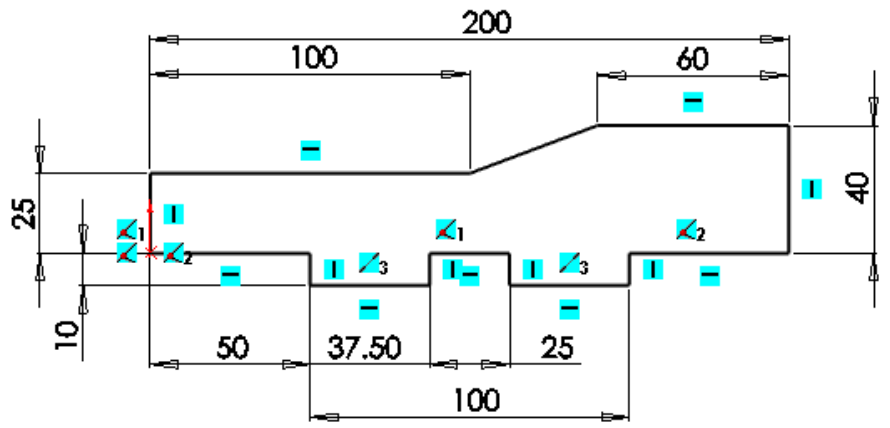
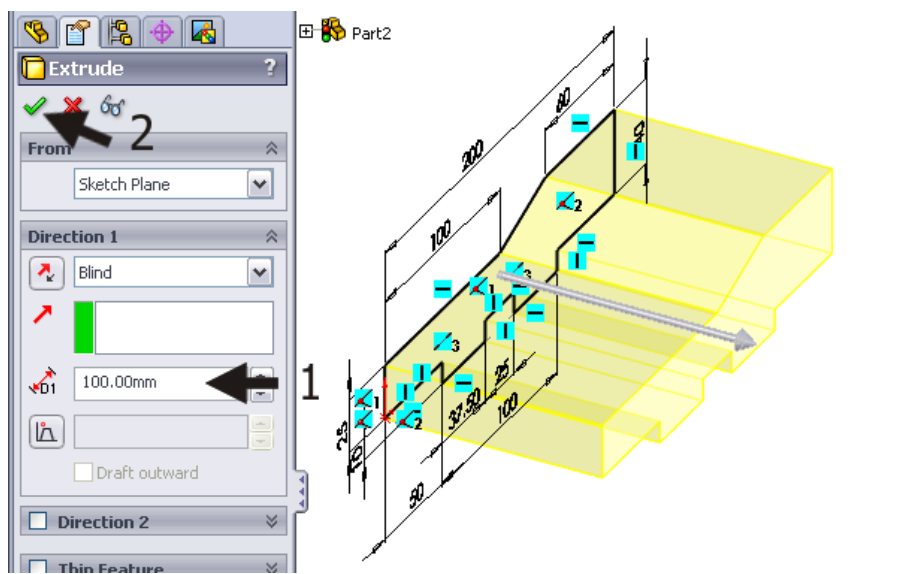
Part material: 6061 Alloy.

Density = 0.0027 g/mm³.

What is the overall mass of the part in grams?

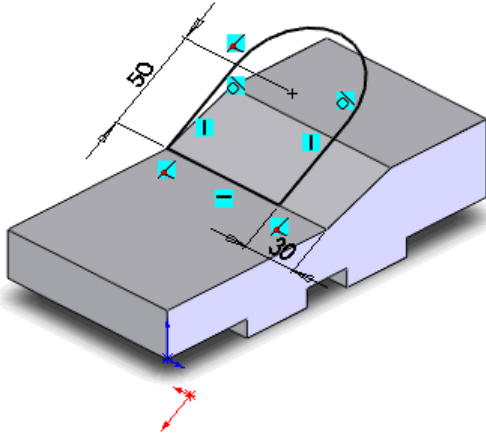
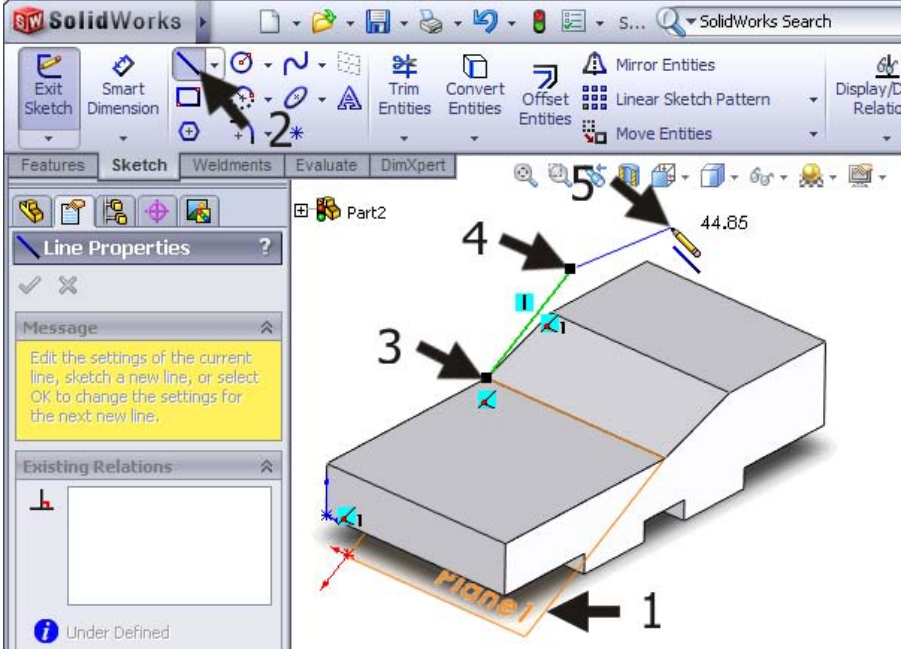
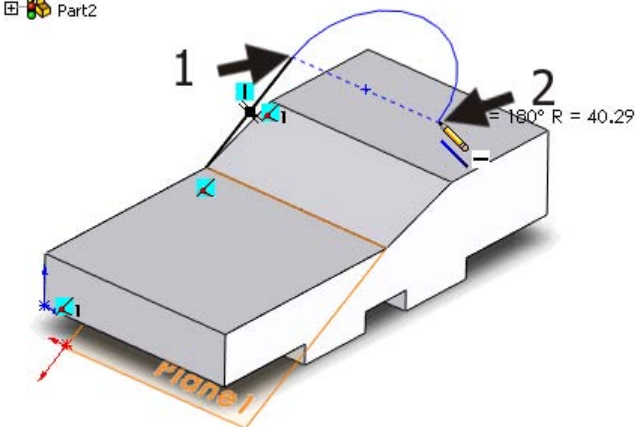
- a. 2040.57
- b. 2004.57
- c. 102.63
- d. 1561.23

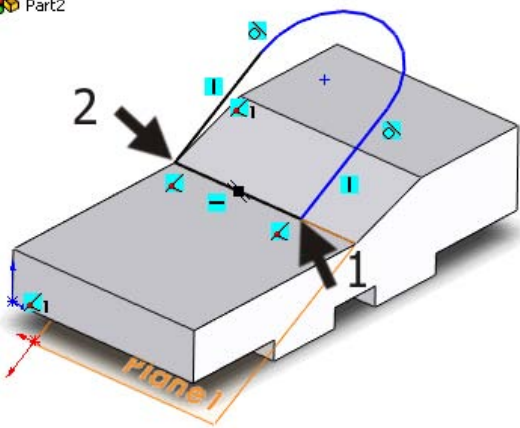
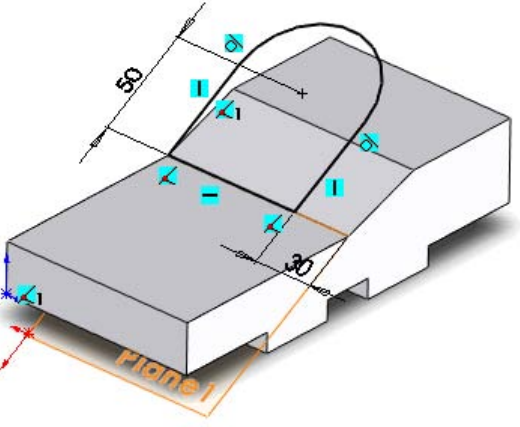
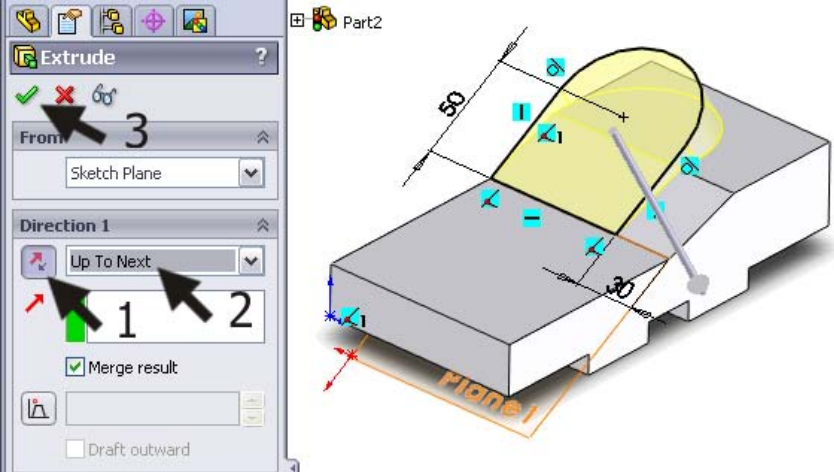


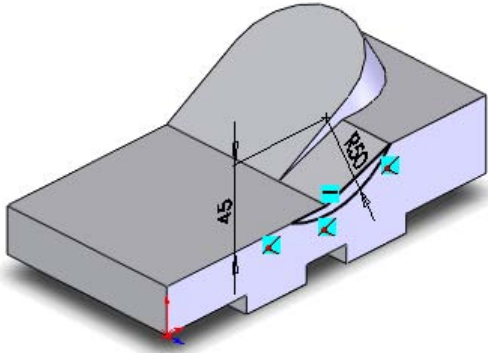
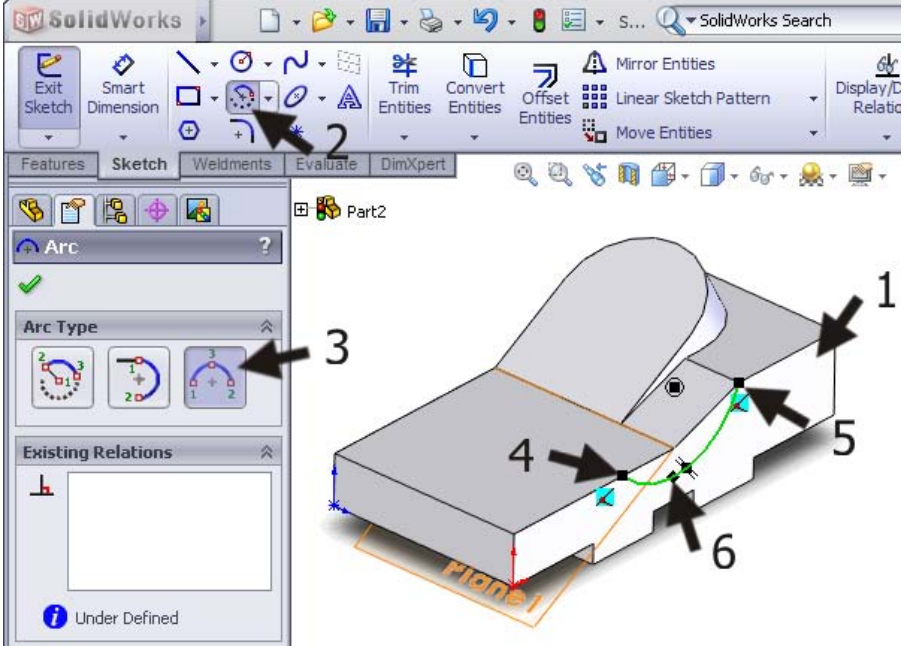
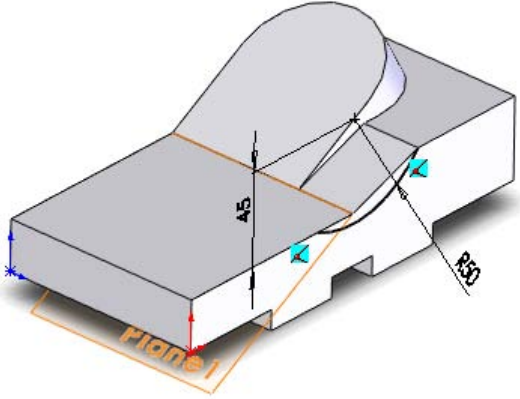
<p>Work plan</p>	<p>Again, you have to think about the way you are going to build this model. Below are the steps you should take. Every step is a feature.</p> 
<p>49 Open a new part and make the sketch as shown on the right on the Right Plane.</p>	
<p>50 Extrude the sketch to '100mm'.</p>	

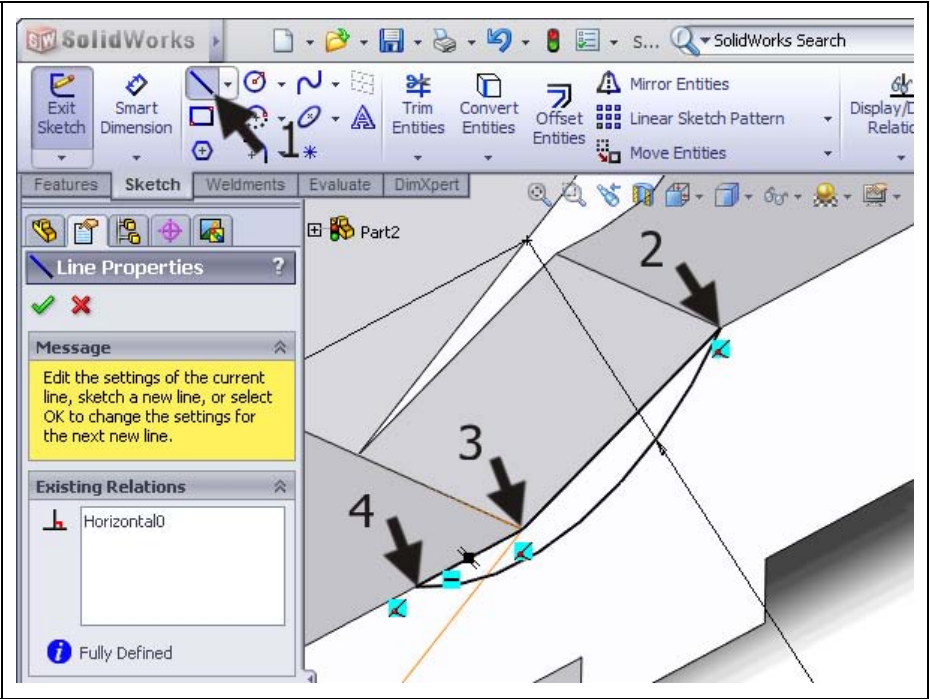
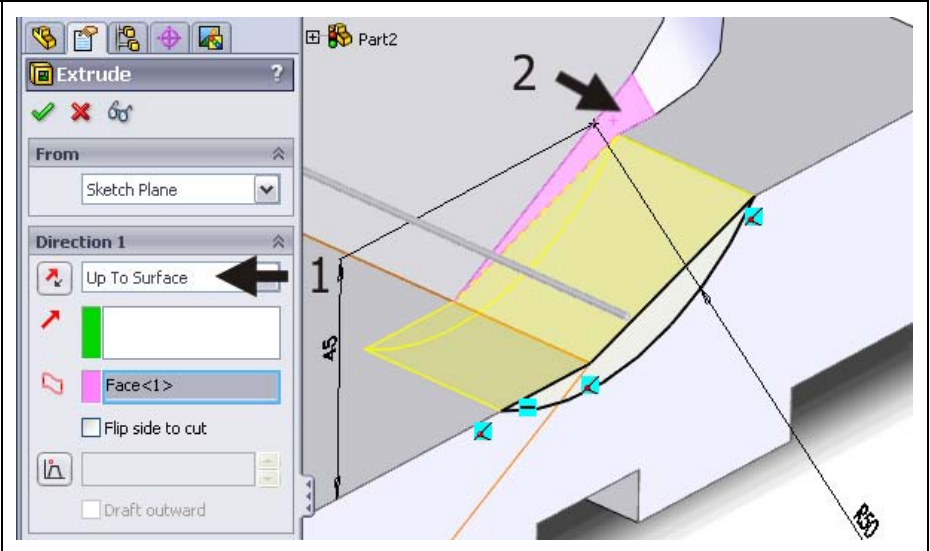
<p>51 We will create the first auxiliary plane:</p> <ol style="list-style-type: none"> 1. Select the edge as shown. 2. Hold the <Ctrl> key and select the plane as shown in the illustration. 3. Click on the arrow beneath 'Reference Geometry' in the CommandManager. 4. Click on 'Plane'. 	
---	--

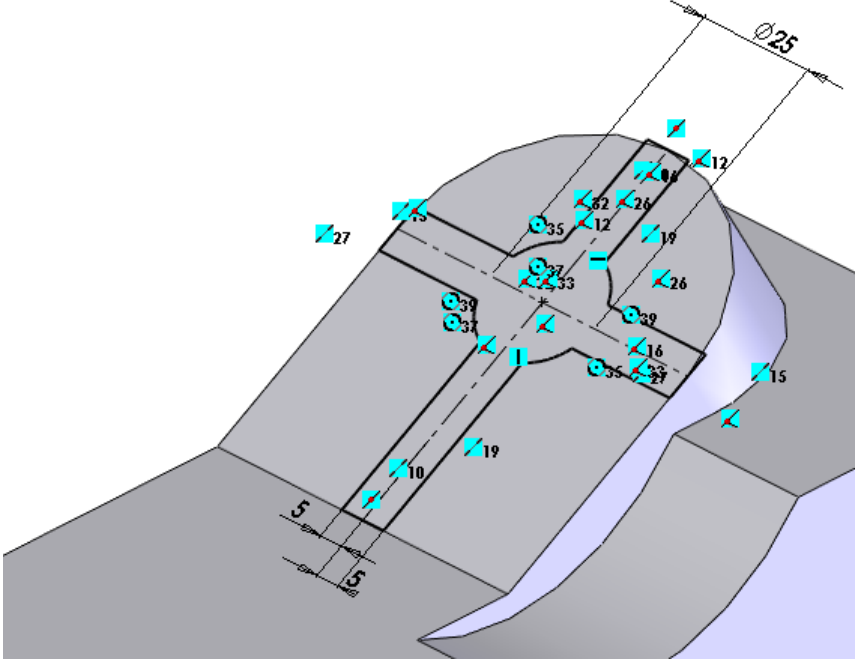
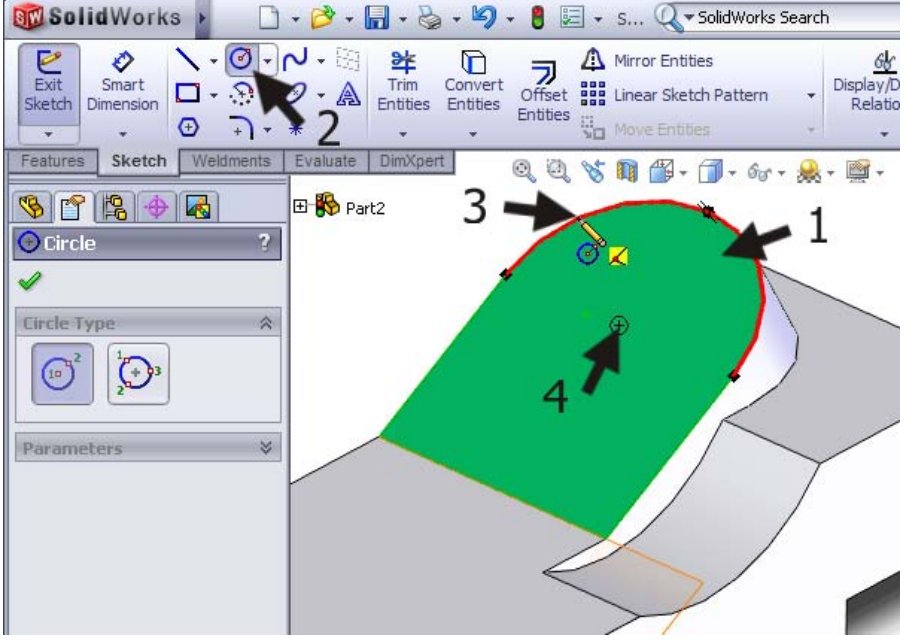
<p>52</p> <ol style="list-style-type: none"> 1. Set the corner of the new plane to '30°' in the PropertyManager. 2. Check 'Reverse direction'. 3. Click on OK. 	
--	--

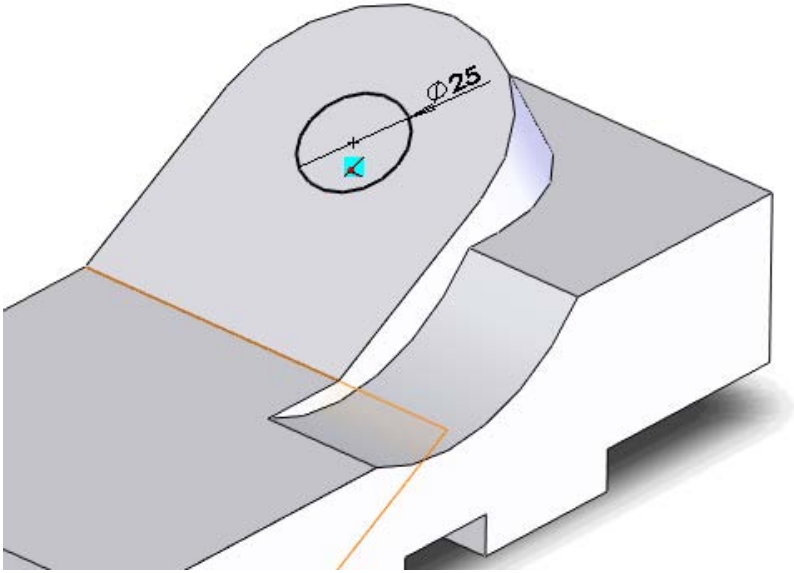
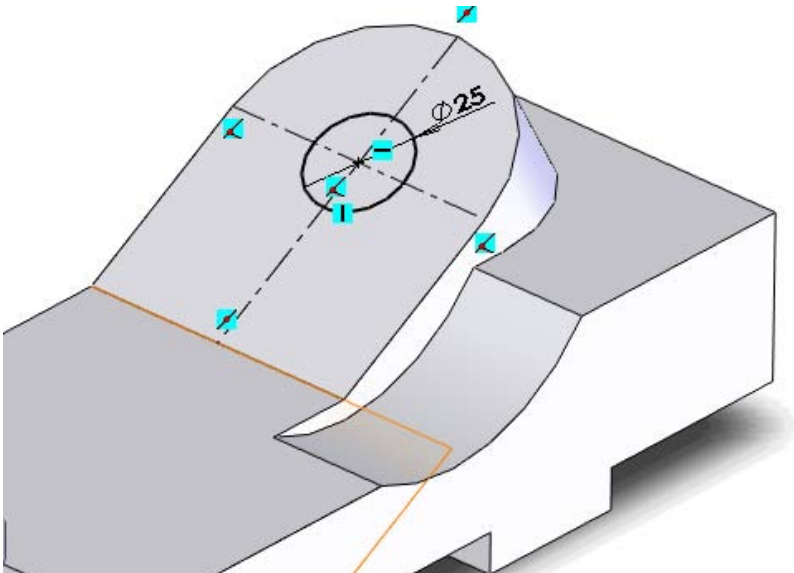
<p>53 Make the sketch as shown in the illustration on the right and continue to Step 58.</p> <p>If you cannot make this sketch by yourself, then follow the next few steps.</p>	
<p>54</p> <ol style="list-style-type: none"> 1. Select the auxiliary plane you have just created. 2. Click on Line in the CommandManager. 3. Click as shown to get the beginning of the line. 4. Click as shown to get the second point from the line. 5. Move the cursor away from the last point but do NOT click! 	
<p>55</p> <ol style="list-style-type: none"> 1. Return to the end point of the line with the cursor (do NOT click!) 2. SolidWorks starts drawing an arc now. 3. Click as shown to get the second point of the arc. Make sure to draw half a circle. 	

<p>56</p>	<p>SolidWorks will automatically draw lines again. Draw the two last lines.</p>	
	<p>Tip!</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>57</p>	<p>Set the two dimensions as shown with Smart Dimension.</p>	
<p>58</p>	<p>Make an extrusion from this sketch.</p> <ol style="list-style-type: none"> 1. Click on Reverse Direction in the Property-Manager to make sure that the extrusion goes downwards and not upwards. 2. Select 'Up to Next' to set the depth. 3. Click on OK. 	

<p>59</p>	<p>Make the sketch as shown in the illustration on the right and continue to Step 63.</p> <p>If you cannot make this sketch by yourself, then follow the next few steps.</p>	
<p>60</p>	<ol style="list-style-type: none"> 1. Select the plane to make a sketch on. 2. Click on Arc in the CommandManager. 3. Click on 3 Point Arc in the PropertyManager. 4. Set the first arc point at the corner as shown. 5. Set the second point on the edge. 6. Set the third point at a random position. 	
<p>61</p>	<p>Insert the two dimensions as shown.</p>	

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

<p>64 Make the sketch on the sloped plane as shown in the illustration on the right and continue to Step 58.</p> <p>If you cannot make this sketch by yourself, then follow the next few steps.</p>	
<p>65</p> <ol style="list-style-type: none"> 1. Select the sloped plane first to make a sketch on. 2. Click on Circle in the CommandManager. 3. Keep the cursor still just above the rounded edge at the top of the plane. Do NOT click! 4. The midpoint of the edge appears. Click on that to set the middle of the circle. 	

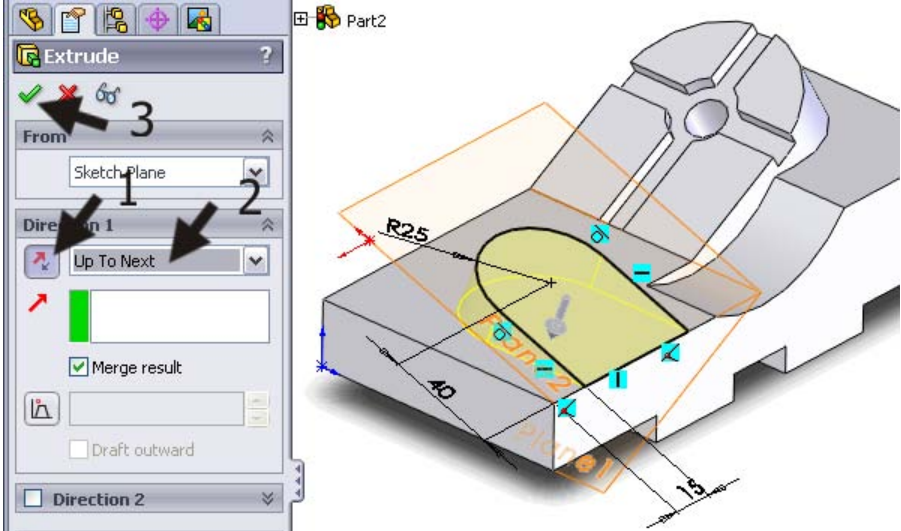
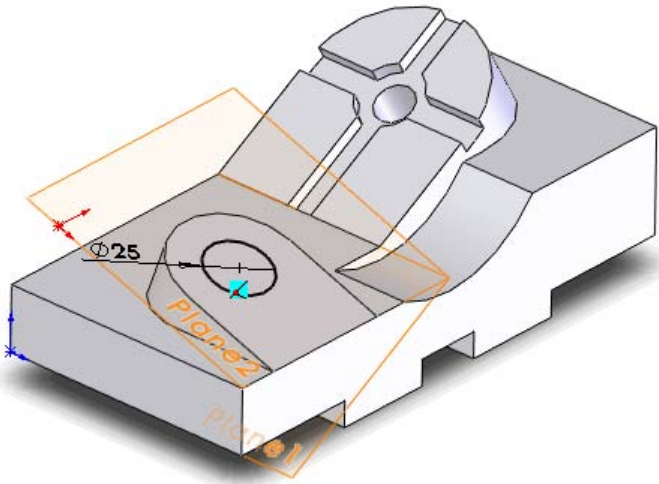
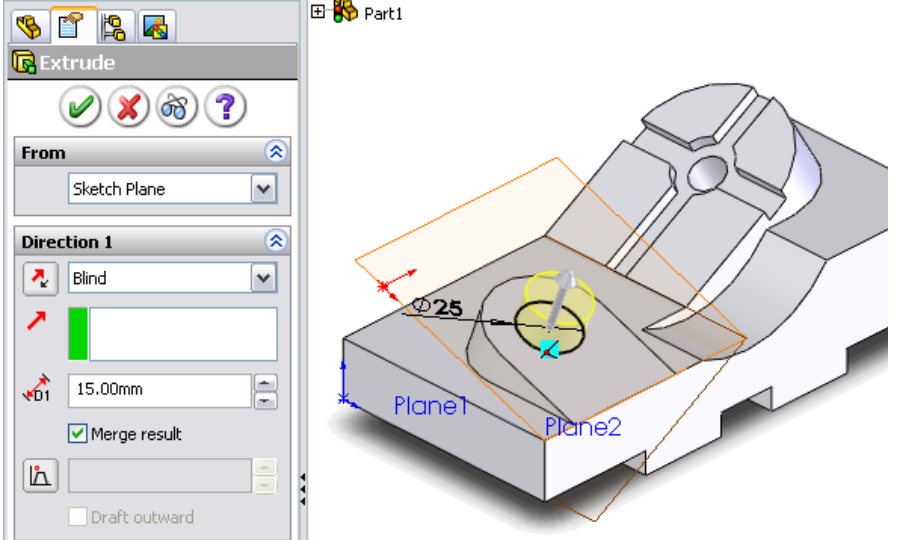
<p>66</p>	<p>Draw the circle and set the dimension.</p>	
<p>67</p>	<p>Draw two centerlines as shown on the right. Push the <Esc> key after you have drawn the first centerline, and then draw the second centerline.</p>	

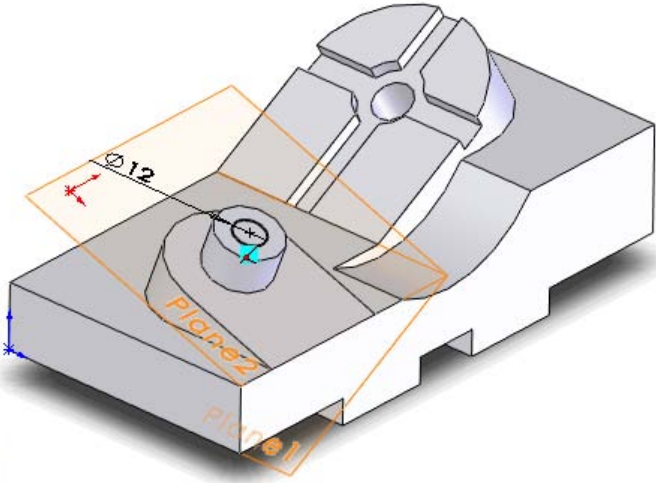
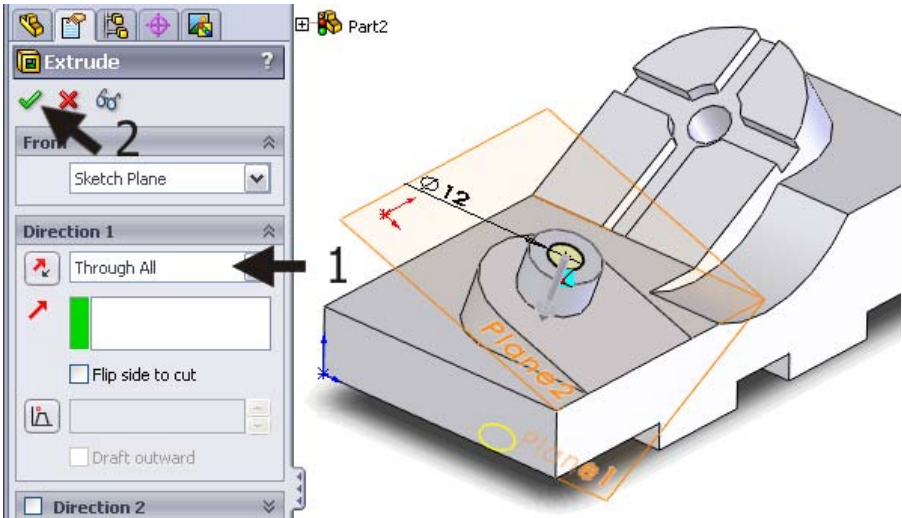
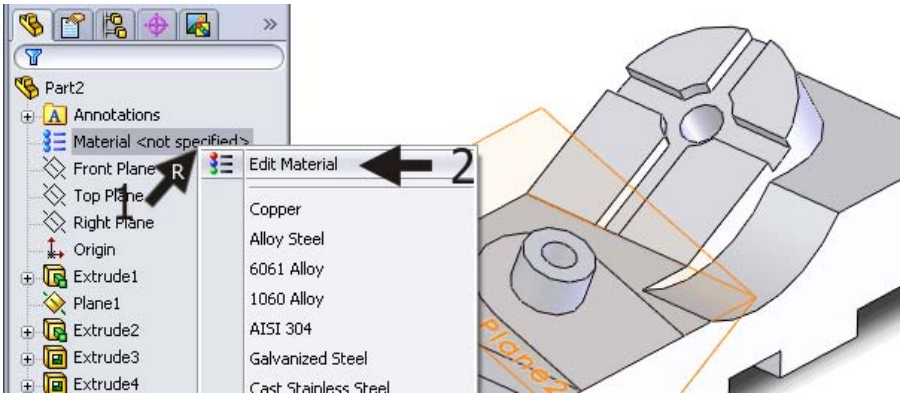
<p>68</p> <ol style="list-style-type: none"> 1,2 Select both centerlines (use the <Ctrl> key). 3. Click on 'Offset Entities' in the CommandMa-nager. 4. The distance is '5mm'. 5. Select the option 'Bi-directional'. 6. Select the option 'Cap ends'. 7. Select the option 'Lines'. 8. Click on OK. 	<p>The screenshot shows the SolidWorks interface with the 'Offset Entities' dialog box open. The dialog box has a 'Parameters' section with the following options: 'Add dimensions' (checked), 'Reverse' (unchecked), 'Select chain' (unchecked), 'Bi-directional' (checked), 'Make base construction' (unchecked), 'Cap ends' (checked), and 'Arcs' (unchecked). The 'Lines' radio button is selected. The 3D model shows a circular hole with a diameter of 25mm. Four lines are drawn through the hole, and arrows indicate the offset direction. The distance is set to 5.00mm. The 'Offset Entities' button is highlighted in the CommandManager.</p>
<p>69</p> <ol style="list-style-type: none"> 1. Click on 'Trim Entities' in the CommandMa-nager. 2. Click on the option 'Trim away inside' in the PropertyManager. 3. Click on the circle. 4. Click on all four lines that run through the circle. The pieces at the inside of the circle will be removed. 	<p>The screenshot shows the SolidWorks interface with the 'Trim' dialog box open. The dialog box has a 'Message' section and an 'Options' section with the following options: 'Power trim' (unchecked), 'Cap' (unchecked), 'Trim away inside' (checked), 'Trim away outside' (unchecked), and 'Trim to closest' (unchecked). The 3D model shows the same circular hole as in the previous screenshot. A red circle is drawn around the hole, and four lines are drawn through it. Arrows indicate the selection of the circle and the four lines. The 'Trim' button is highlighted in the CommandManager.</p>

<p>70</p>	<ol style="list-style-type: none"> 1. Click on 'Trim to closest' in the PropertyManager. 2. Click on the parts of the circle that you want to be removed. 	
<p>71</p>	<p>Did you trim everything? Now you can make an Extruded Cut from the sketch. Set the depth to '5mm'.</p>	
<p>72</p>	<p>Make the sketch as shown in the illustration on the right.</p>	

<p>73</p>	<p>Make an Extruded Cut 'Through All'.</p>	
<p>74</p>	<p>We will now make the second auxiliary plane.</p> <ol style="list-style-type: none"> 1. Select the edge as shown. 2. Hold the <Ctrl>-key and select the plane as shown in the illustration. 3. Click on the arrow beneath 'Reference Geometry' in the CommandManager. 4. Click on 'Plane'. 	

<p>75</p>	<ol style="list-style-type: none"> 1. Set the angle of the new plane to '20°' in the PropertyManager. 2. Click on Reverse direction, so the plane extends in the right direction. 3. Click on OK. 	
<p>76</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>Tip!</p>	<p>The bottom corner points from the sketch are not exactly on the edge of the model (not coincident). This is because the plane you have inserted (Plane2) is also exactly on that edge. How can you solve this? Hide the plane temporarily. Click on Hide/Show Items, and next on Planes.</p>	

<p>77</p> <p>Make an extrusion from the sketch.</p> <ol style="list-style-type: none"> 1. First click on Reverse Direction in the PropertyManager to extend the extrusion downwards. 2. Select the option 'Up To Next'. 3. Click on OK. 	
<p>78</p> <p>Make the sketch as shown on the right.</p>	
<p>79</p> <p>Extrude the sketch with a height of '15mm'.</p>	

<p>80</p>	<p>Make the sketch as shown on the right.</p>	
<p>81</p>	<p>Make an Extruded Cut 'Through All' from this sketch.</p>	
<p>82</p>	<p>The model is now finished. We will select the kind of material now.</p> <ol style="list-style-type: none"> 1. Right-click on 'Material' in the FeatureManager. 2. Click on 'Edit Material'. 	

<p>83</p> <ol style="list-style-type: none"> 1. Open the list of 'Aluminum Alloys' in the PropertyManager. 2. Select '6061 Alloy'. 3. Verify that the density is the same as the one in the assignment. 4. Click on OK. 	<ol style="list-style-type: none"> 1. Open the list of 'Aluminum Alloys' in the PropertyManager. 2. Select '6061 Alloy'. 3. Verify that the density is the same as the one in the assignment. 4. Click on OK. 	
<p>84</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 pop-up menu you can read a weight of 2040.57 grams. So this is answer A 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. The center of mass in the model itself is also indicated.</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 pop-up menu you can read a weight of 2040.57 grams. So this is answer A 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. The center of mass in the model itself is also indicated.</p>	