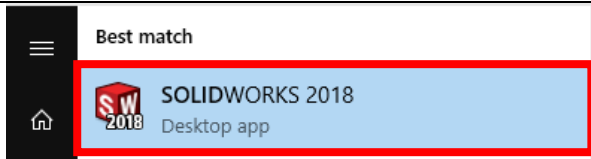

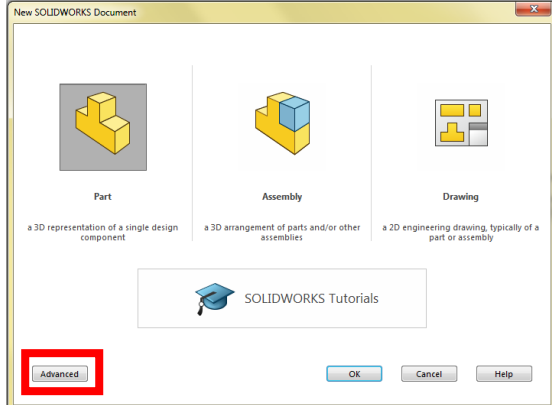
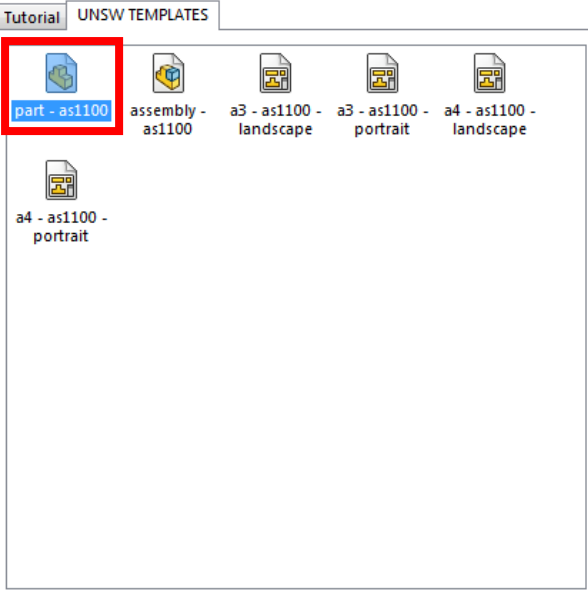
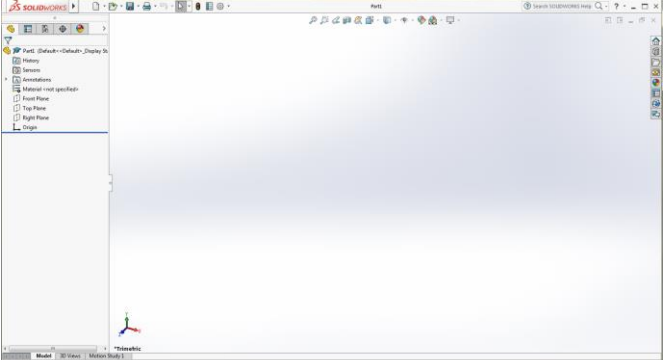


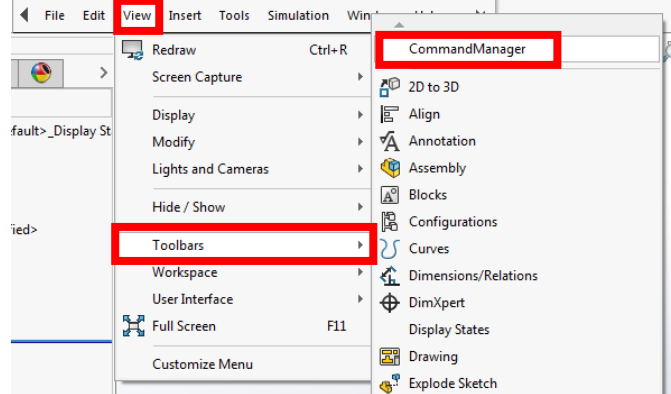
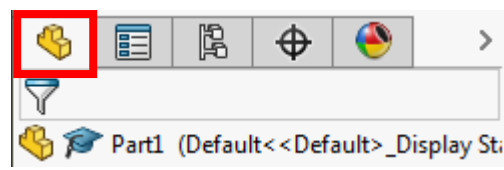
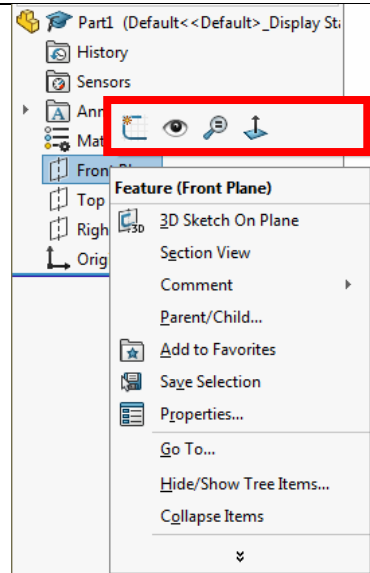
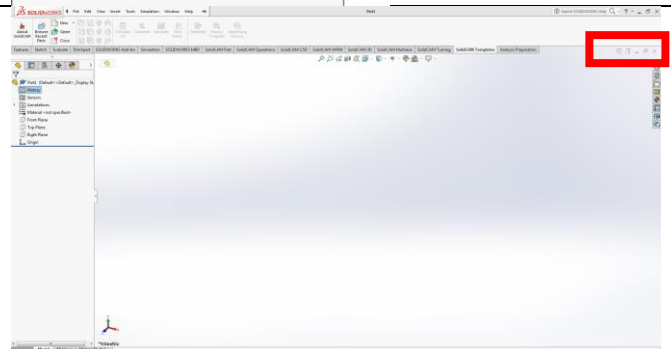
1 – 2D SKETCHING

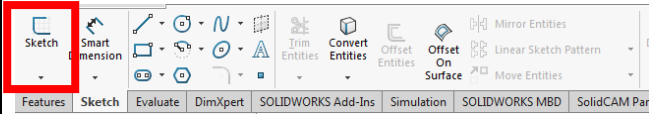
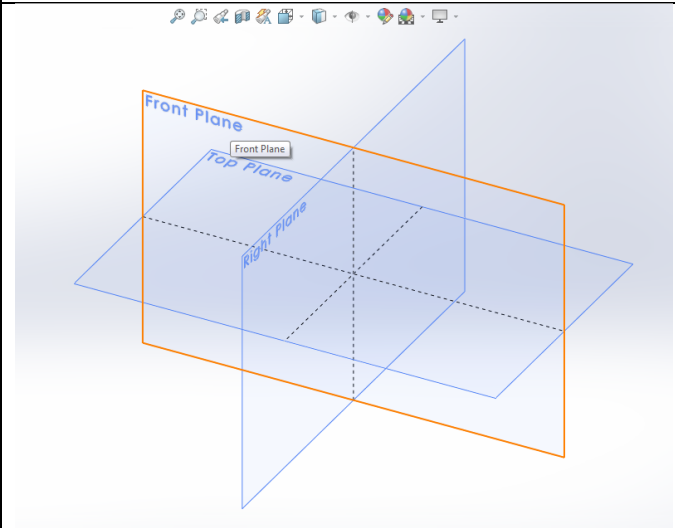
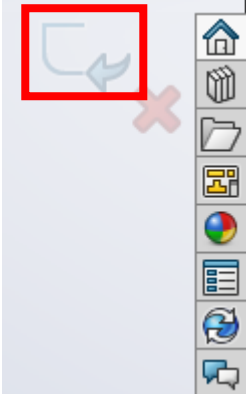
By following this tutorial, you will learn how to:

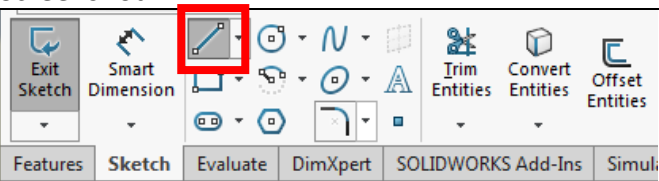
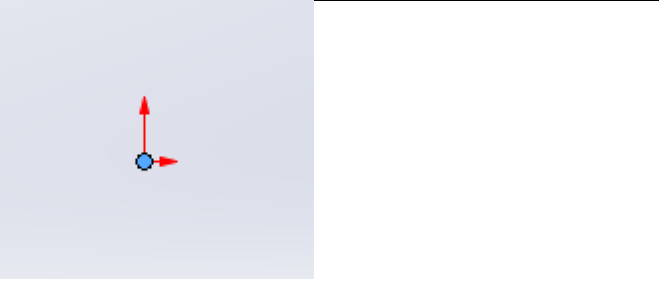
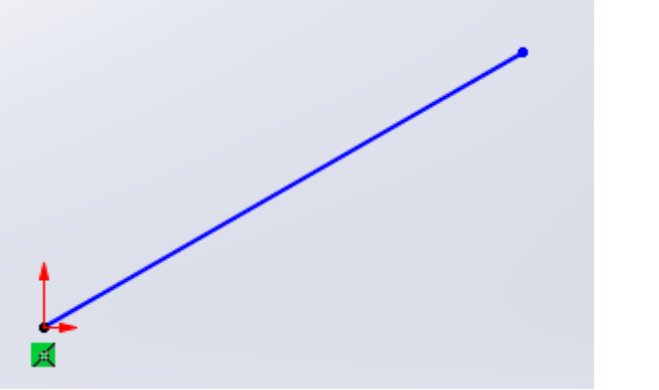
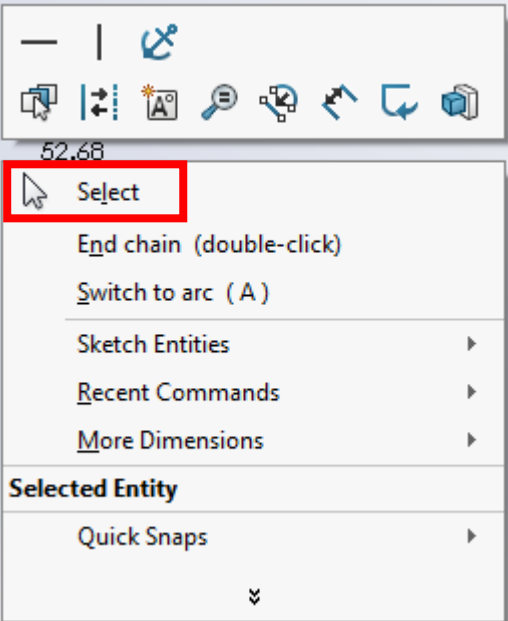
1. Set up a SolidWorks part file
2. Use the SolidWorks menu
3. Create a new sketch plane
4. Create straight lines, circles, rectangles, and slots
5. Fully dimension shapes
6. Use the trim tool
7. Use a sketch fillet
8. Create a relation between two entities

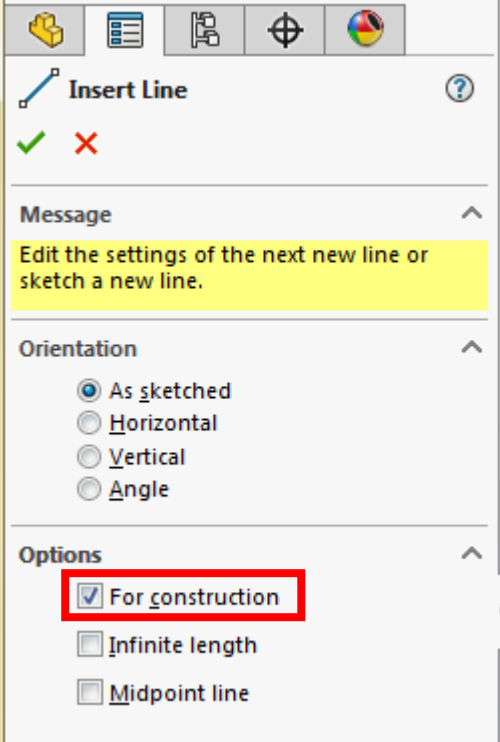
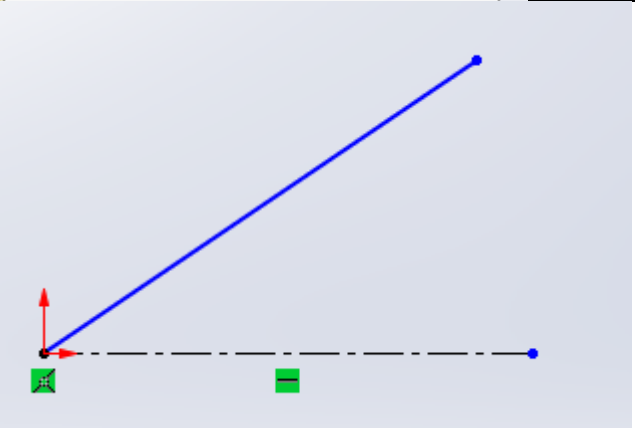
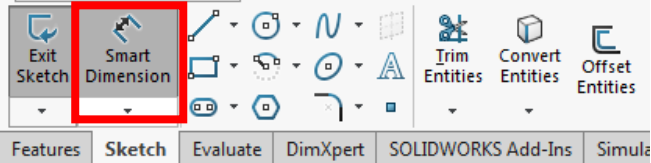
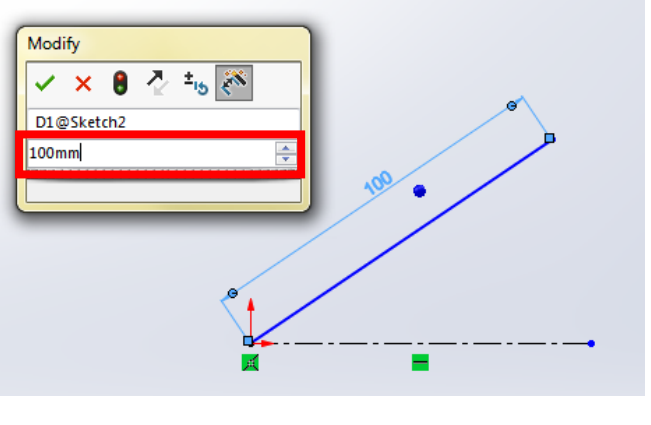
Set up a SolidWorks part file		
No.	Instruction	Screenshot
1	Open SolidWorks from the start menu.	
2	Click on the New File button.	
3	Click on the Advanced button.	

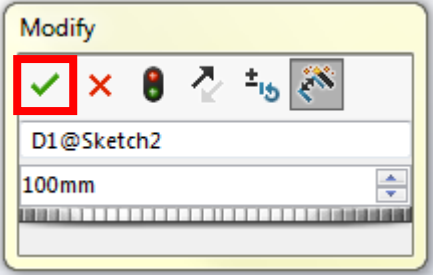
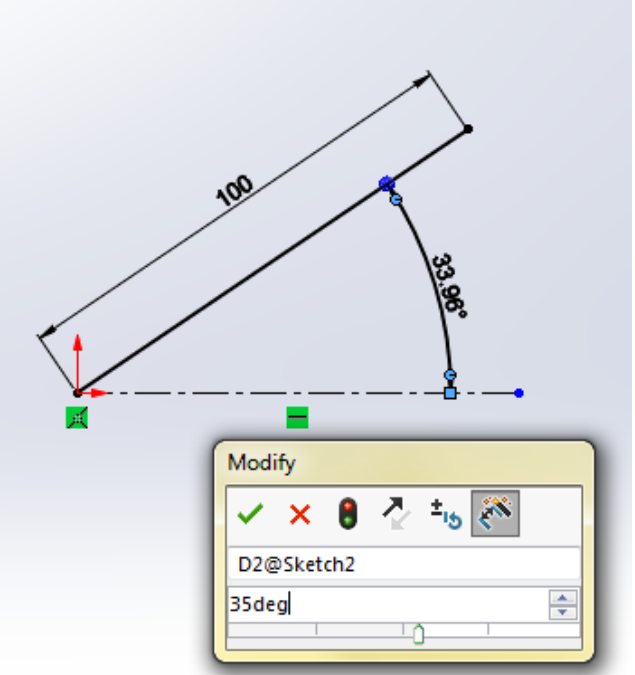
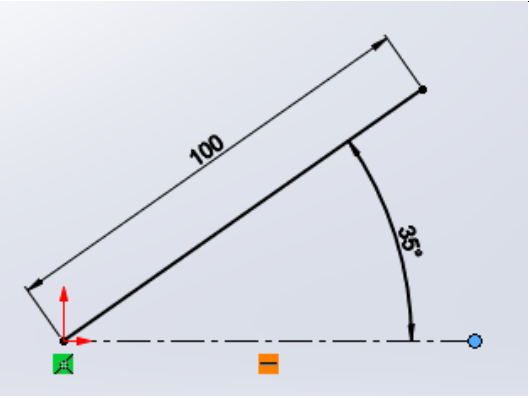
4	<p>Click on the “part - as1100” option under the UNSW TEMPLATES tab.</p> <p>Click OK.</p>	
5	<p>You have now created a new part file.</p>	

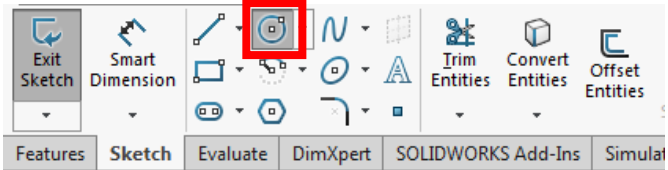
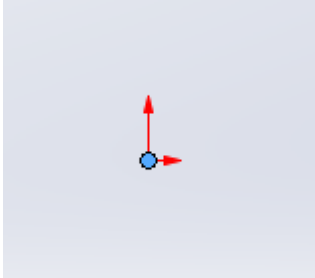
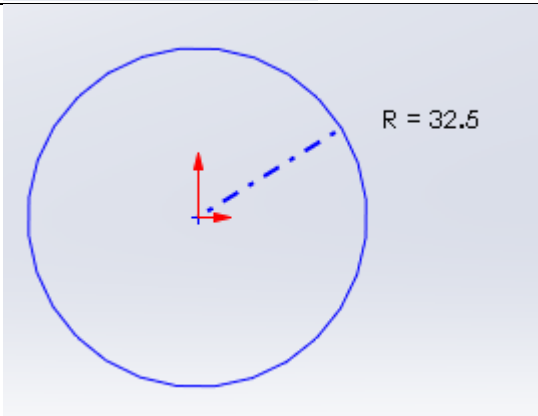
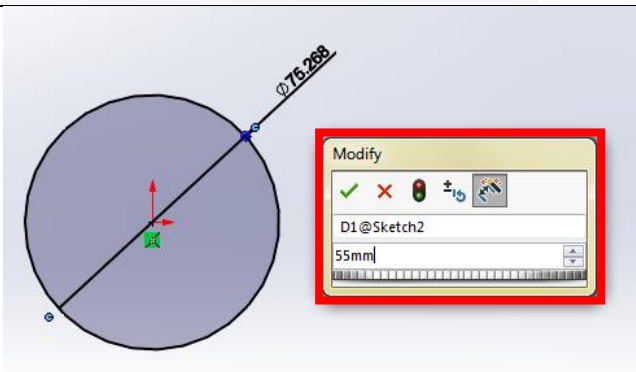
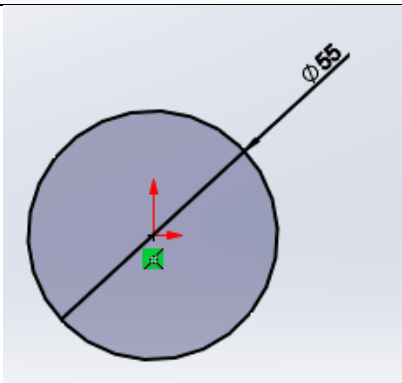
Use the SolidWorks menu		
No.	Instruction	Screenshot
1	On the menu bar, go to View >> Toolbars >> CommandManager to bring up the command manager	
2	On the left sidebar, ensure that the feature manager is selected	
3	Right click on an item in the tree to see options. Hover your mouse over the icons for a description.	
4	Click on the minimise option on the work area to minimise the current part file.	

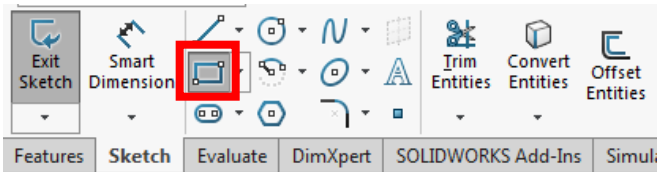
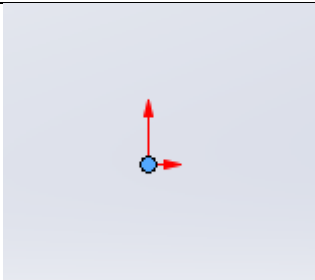
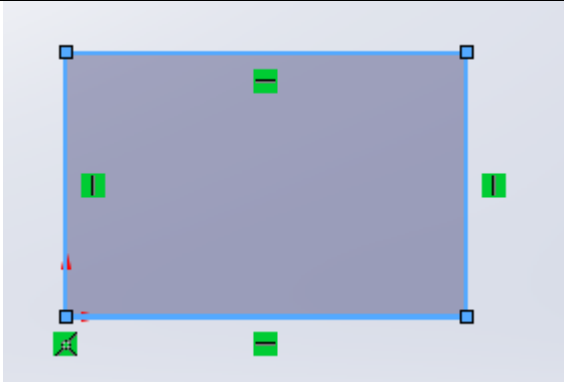
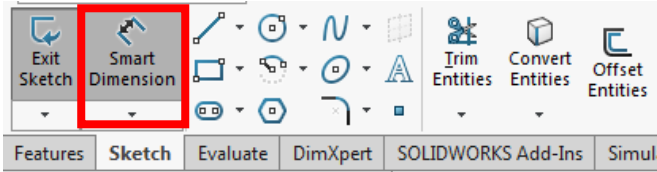
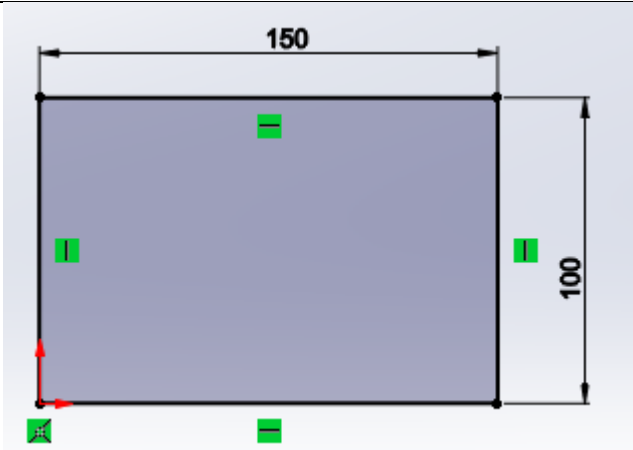
Create a new sketch plane		
No.	Instruction	Screenshot
1	Go to the Sketch tab on the command manager. Click "Sketch".	
2	Click on the appropriate plane.	
3	Click the Exit Sketch button to exit the sketch plane.	

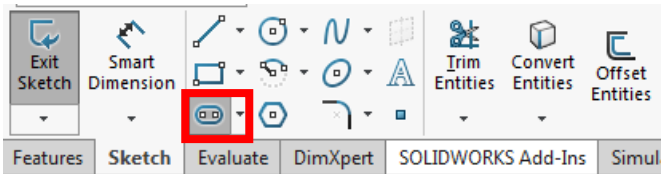
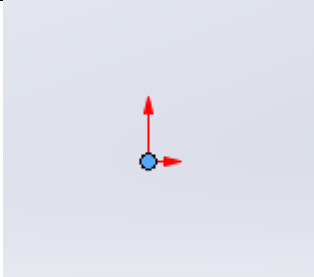
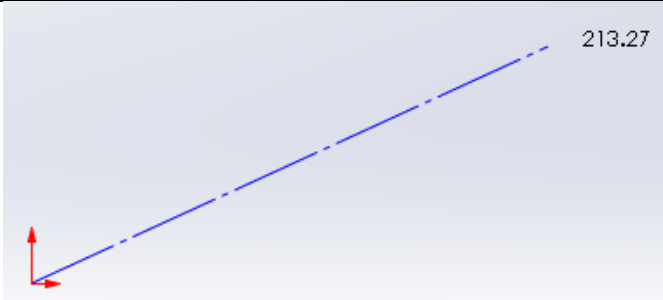
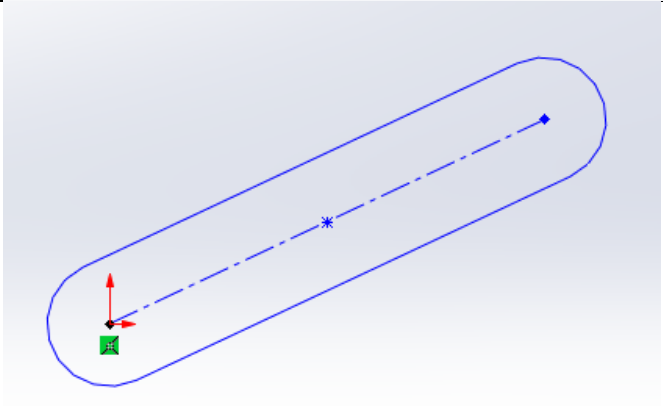
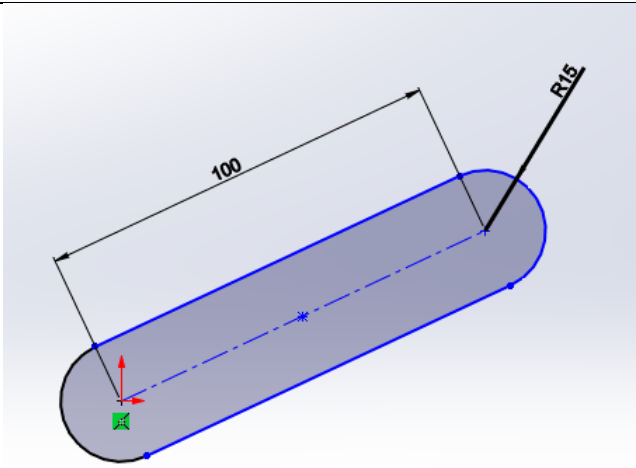
Create a line		
No.	Instruction	Screenshot
1	Click the "Line" button in the Sketch tab.	
2	Click on the origin to specify the starting point of the line	
3	Click on the desired ending point of the line.	
4	Right click and choose "Select" to finish the drawing action.	

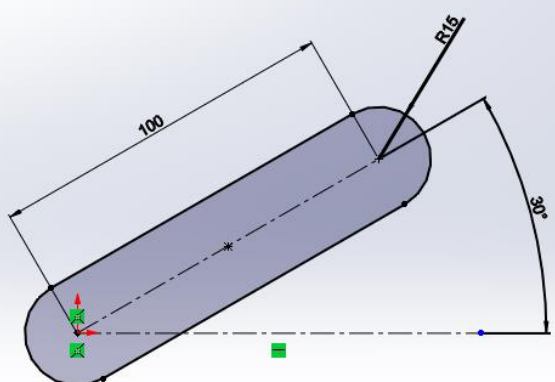
5	To create a construction line, choose the “For Construction” option.	 <p>The image shows the 'Insert Line' dialog box in SolidWorks. The 'Orientation' section has 'As sketched' selected. The 'Options' section has 'For construction' checked, which is highlighted with a red rectangle. Other options like 'Infinite length' and 'Midpoint line' are unchecked.</p>
6	Create a horizontal reference line.	 <p>The image shows a sketch in SolidWorks. A solid blue line is drawn at an angle. A dashed horizontal line is drawn below it, serving as a reference. A red arrow points to the start of the dashed line.</p>
7	Click on “Smart Dimension” in the Sketch tab.	 <p>The image shows the 'Sketch' tab in the SolidWorks ribbon. The 'Smart Dimension' button is highlighted with a red rectangle.</p>
8	Click on the created line and specify a value in the Modify box.	 <p>The image shows the 'Modify' dialog box in SolidWorks. The 'D1@Sketch2' dimension is selected, and the value '100mm' is entered in the text box, which is highlighted with a red rectangle. The background shows a sketch with a dimension line indicating a length of 100.</p>

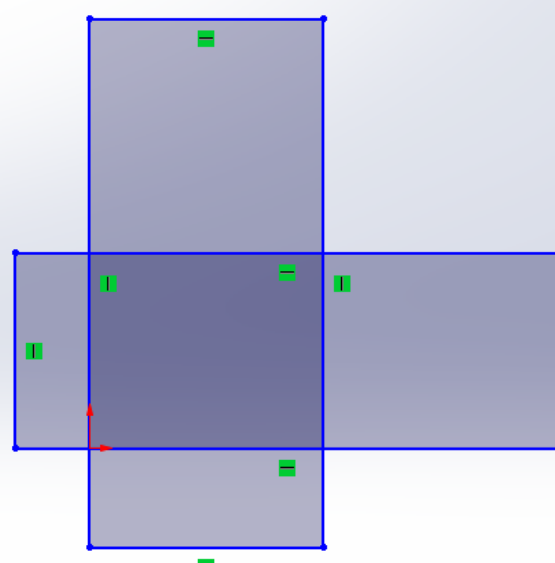
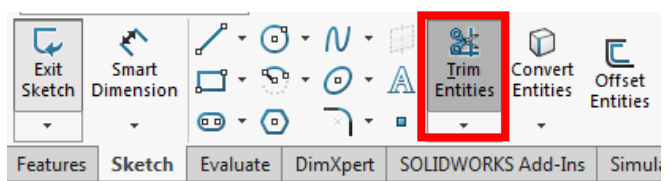
9	Click the OK button to confirm.	
10	Click the Smart Dimension button. Click on both lines and specify a value in the Modify box. Click the OK button to confirm.	
	You have now fully dimensioned the created line.	

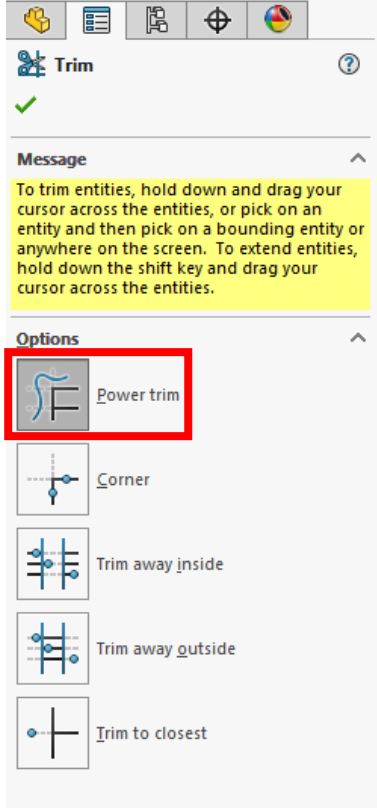
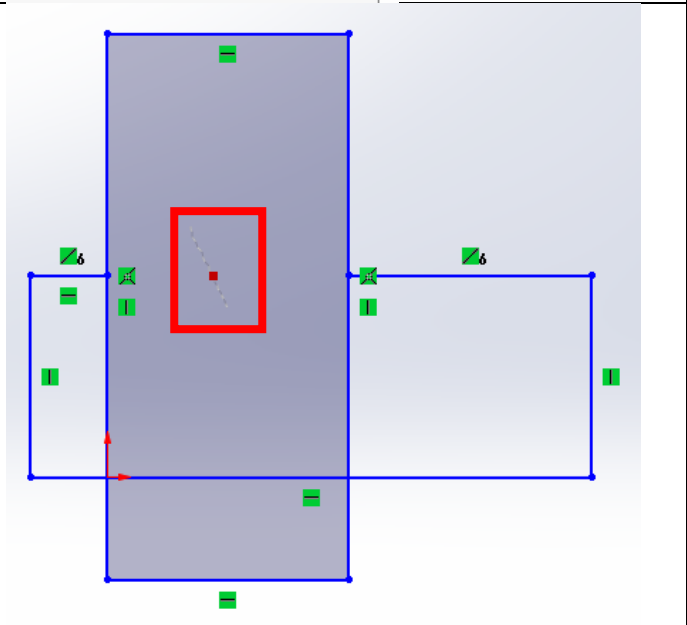
Create a Circle		
No.	Instruction	Screenshot
1	Click the "Circle" button in the Sketch tab.	
2	Click on the origin to specify the starting point of the line	
3	Move the pointer to specify the radius of the circle. Click to set.	
4	Use "Smart Dimension" to specify the dimensions of the circle.	
5	You have now fully dimensioned the circle.	

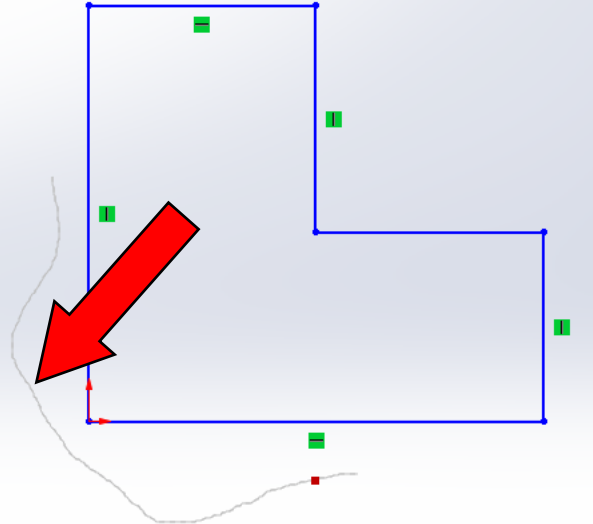
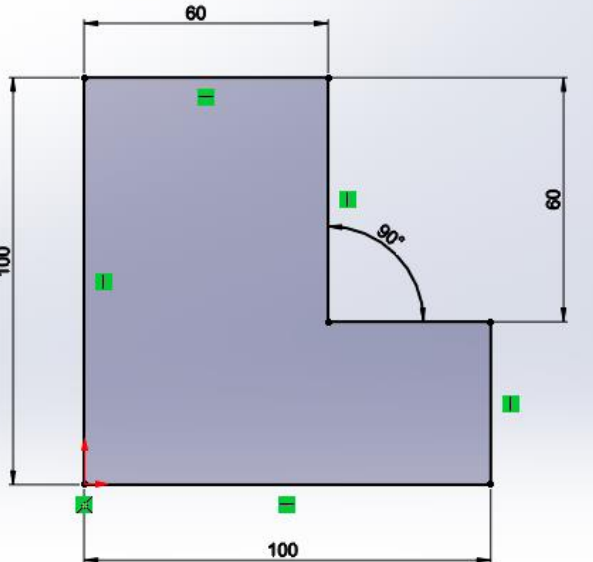
Create a Circle		
No.	Instruction	Screenshot
1	Click the "Rectangle" button in the Sketch tab.	
2	Click on the origin to specify the starting point of the line	
3	Move the pointer to specify the size of the rectangle. Click to set.	
4	Use "Smart Dimension" to specify the dimensions of the rectangle.	
5	You have now fully dimensioned the rectangle.	

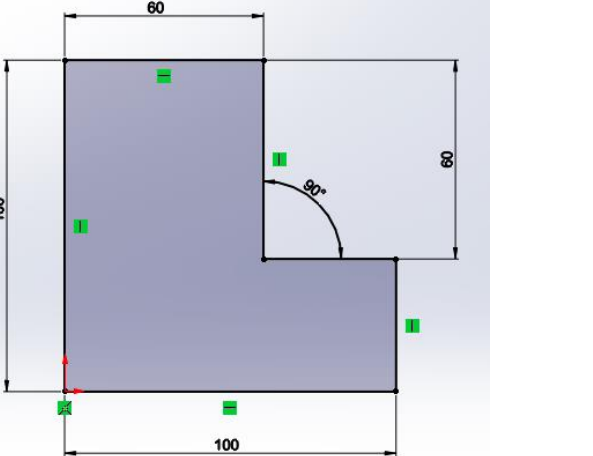
Create a Slot		
No.	Instruction	Screenshot
1	Click the "Straight Slot" button in the Sketch tab.	
2	Click on the origin to specify the starting point of the line	
3	Move the pointer to specify the length of the slot. Click to set.	
4	Move the pointer to specify the width of the slot. Click to set.	
5	Use the "Smart Dimension" tool to dimension the slot size.	

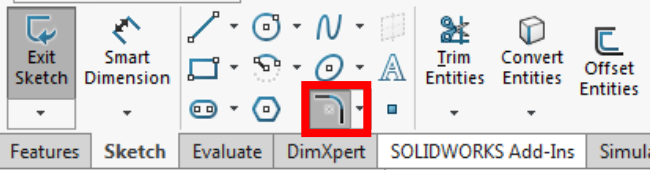
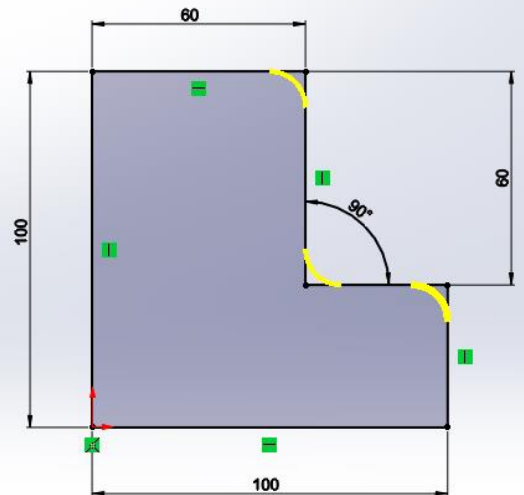
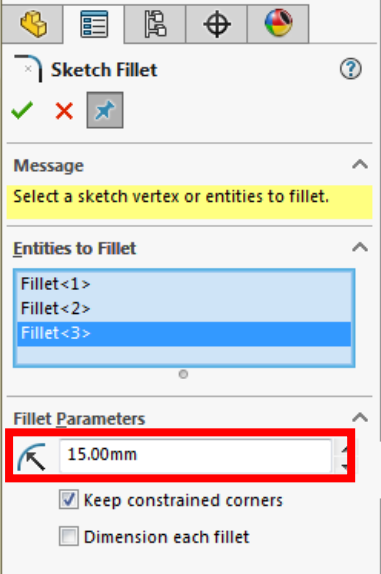
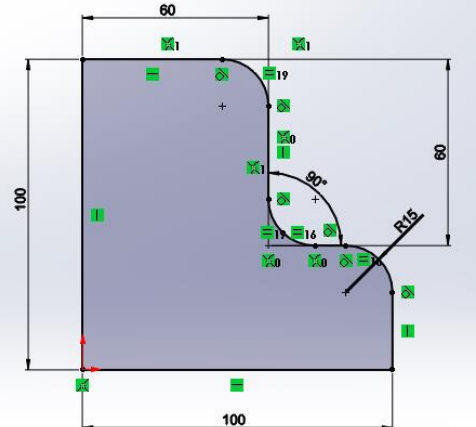
6	Use a construction line with the "Smart Dimension" tool to dimension the slot angle.	
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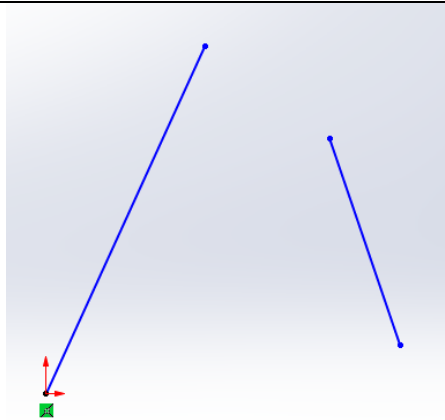

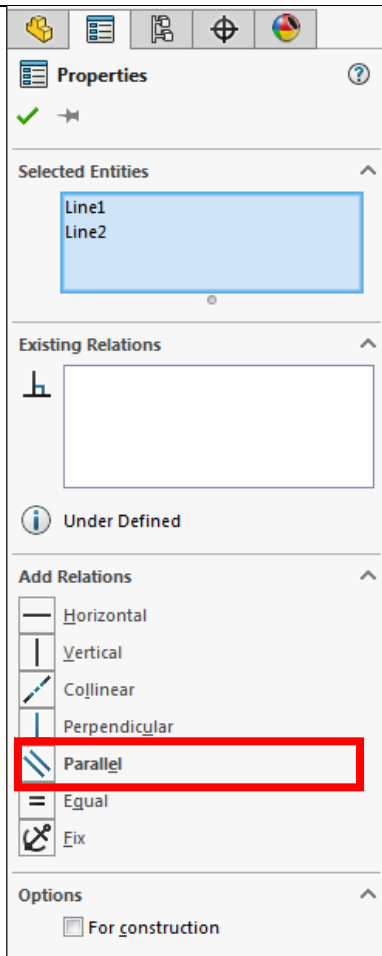
Use the trim tool		
No.	Instruction	Screenshot
1	Create two overlapping rectangles	
2	Select the "Trim Entities" button in the Sketch tab.	

3	Select the “Power Trim” option.	 <p>Trim</p> <p>Message</p> <p>To trim entities, hold down and drag your cursor across the entities, or pick on an entity and then pick on a bounding entity or anywhere on the screen. To extend entities, hold down the shift key and drag your cursor across the entities.</p> <p>Options</p> <p>Power trim</p> <p>Corner</p> <p>Trim away inside</p> <p>Trim away outside</p> <p>Trim to closest</p>
4	Click and drag the pointer across a line to remove it.	

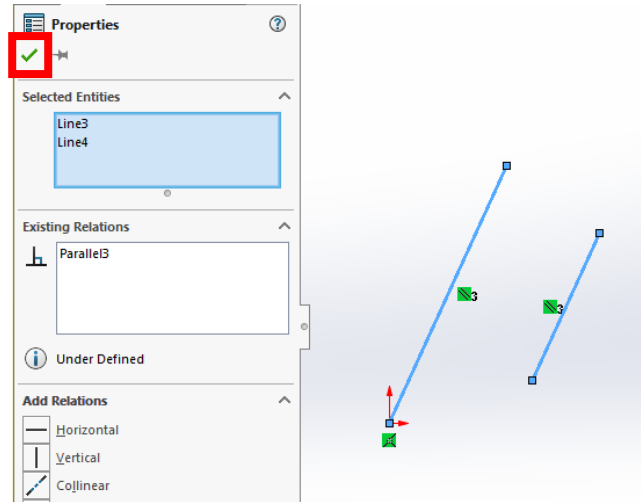
5	<p>Repeat to trim other lines.</p> <p>TIP: You can cross multiple lines at once by holding and dragging across all applicable lines.</p>	
6	<p>Use the “Smart Dimension” tool to fully dimension the shape.</p>	

Use a sketch fillet		
No.	Instruction	Screenshot
1	<p>Use the shape created above.</p>	

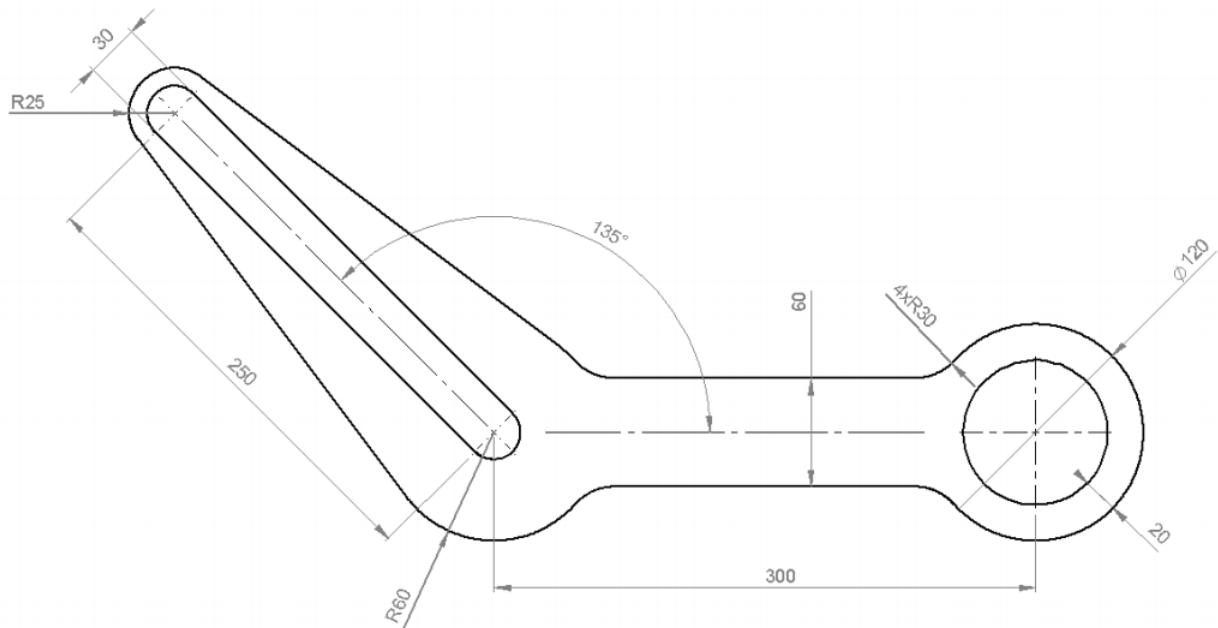
2	Select the "Sketch Fillet" button in the Sketch tab.	
3	Click all the corners you want to fillet	
4	Enter the desired corner radius in the "Fillet Parameters" menu.	
5	Ensure that the shape is fully dimensioned.	

Create a relation between two entities		
No.	Instruction	Screenshot
1	Draw two lines.	
2	<p>Click the first line.</p> <p>Hold CTRL on the keyboard.</p> <p>Click the second line.</p> <p>This will bring up the Relation Properties menu.</p>	
3	Choose the appropriate relation.	

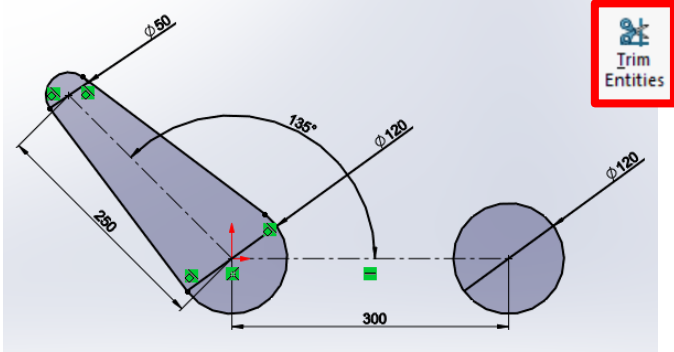
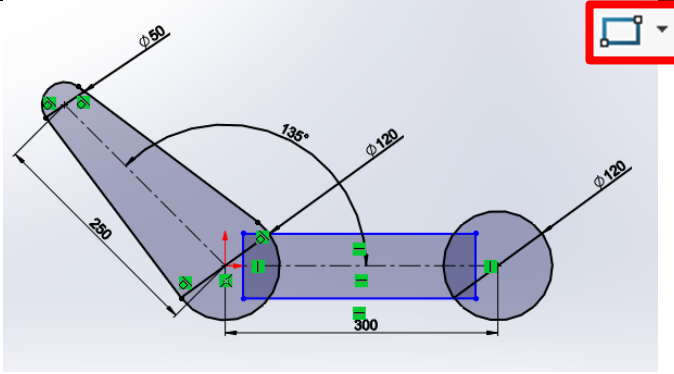
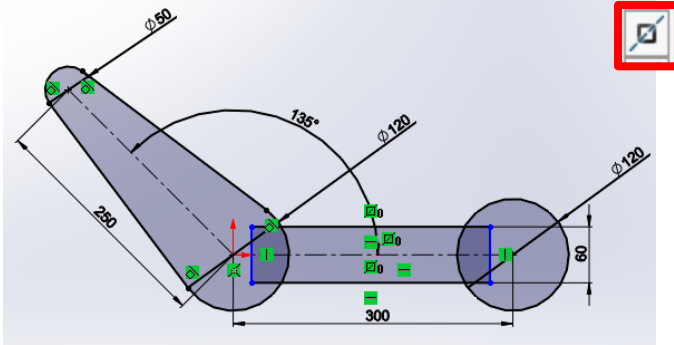
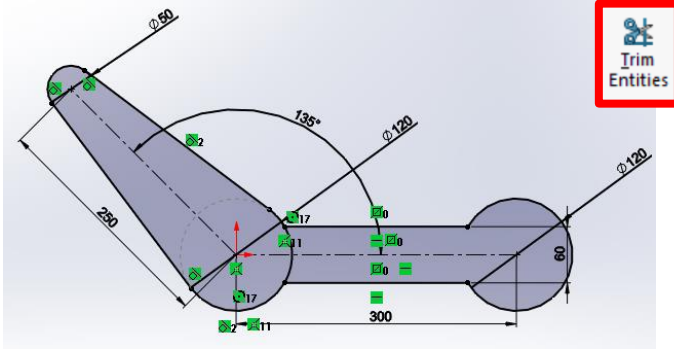
4 Click the OK button to confirm.

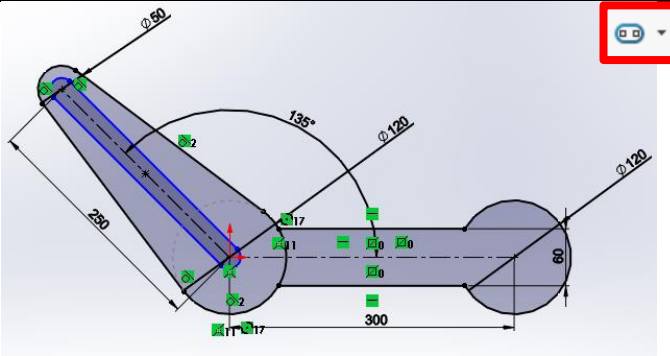
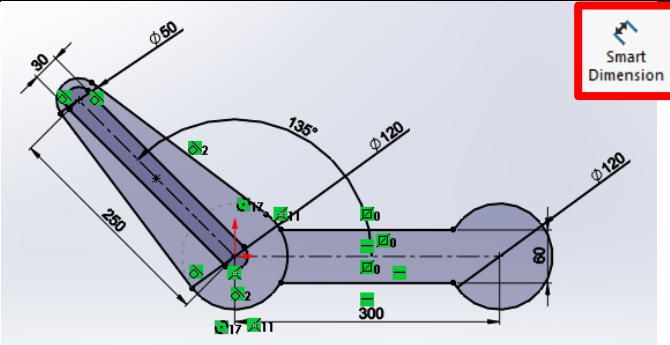
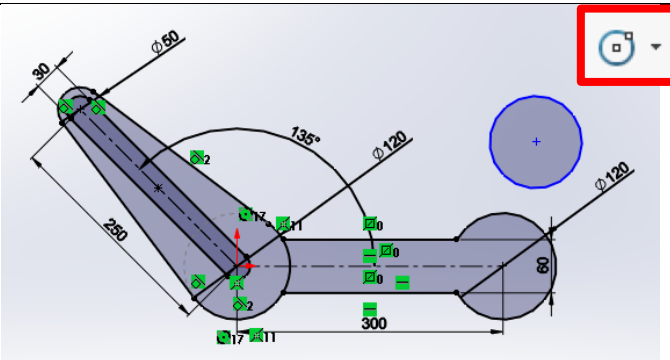

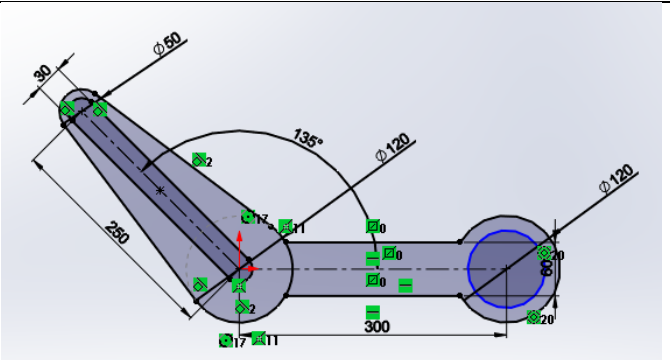


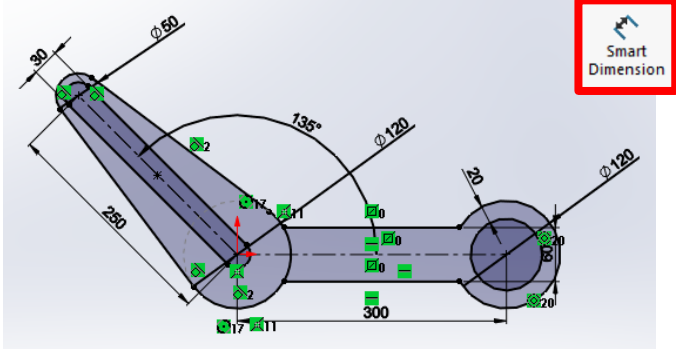
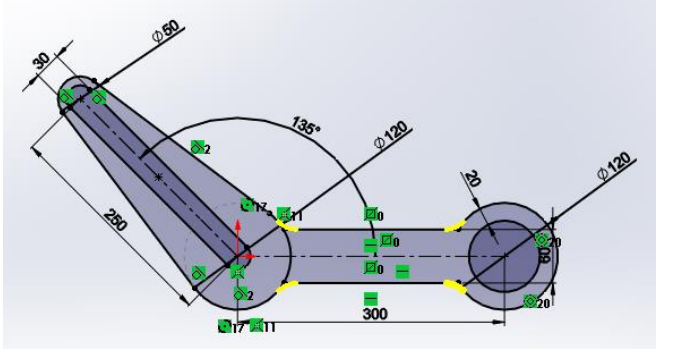
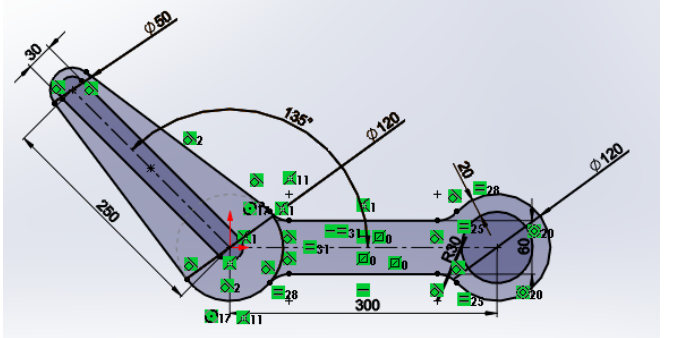
Sketching Example



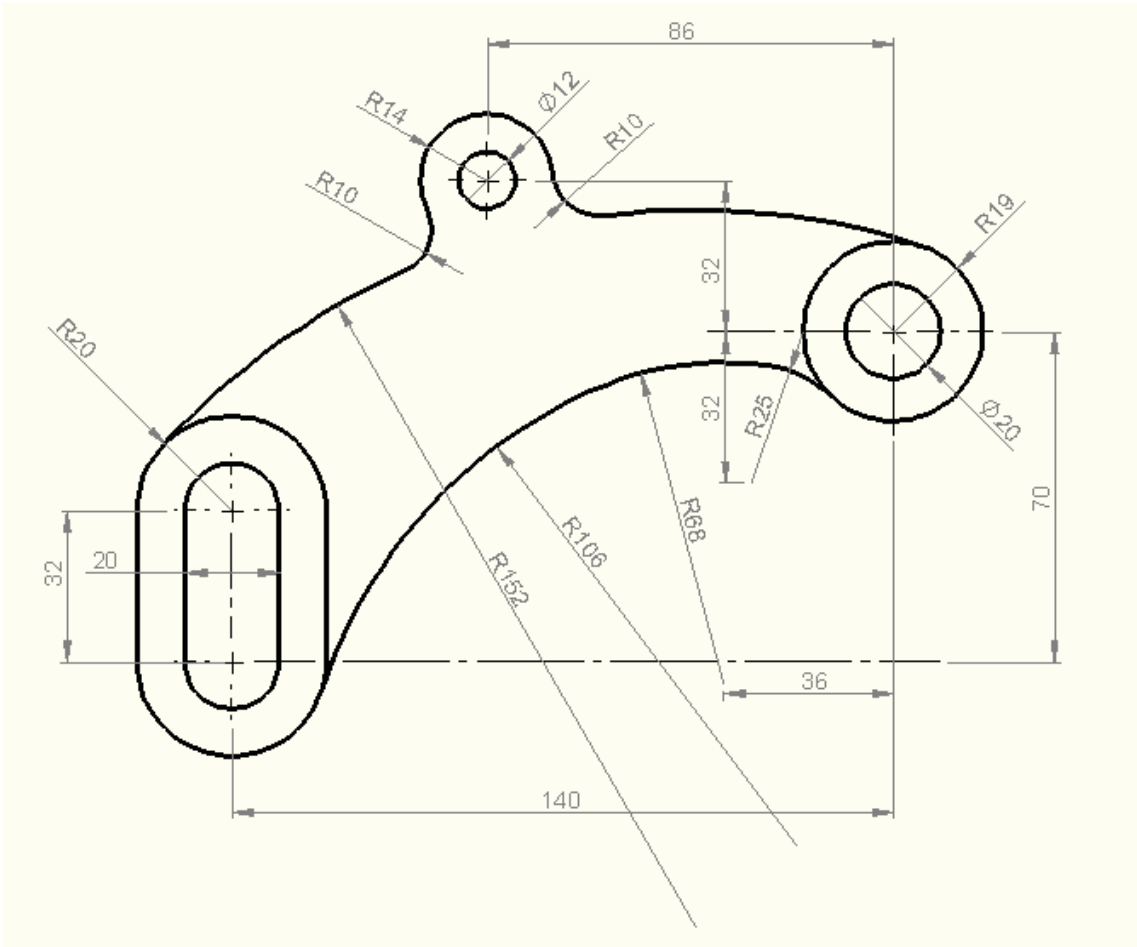
Sketching Example		
No.	Instruction	Screenshot
1	Create two construction lines.	
2	Smart Dimension the size and angle of the two lines. <ul style="list-style-type: none"> - Horizontal line = 300mm - Inclined line = 250mm - Angle = 135° 	

7	Trim all redundant lines.	
8	Create a horizontal rectangle as shown.	
9	Smart dimension the rectangle height 60mm. Use the Symmetry relation to position the rectangle on the centreline.	
10	Trim all redundant lines.	

11	<p>Create a slot using the inclined construction line as a centreline.</p> <p>Ensure that the ends are concentric to the existing circles.</p>	
12	<p>Smart Dimension the slot.</p> <ul style="list-style-type: none"> - Width = 30mm 	
13	<p>Create a new circle close to the right-most circle.</p>	
14	<p>Add a Concentric relation between the two circles.</p> 	

15	Smart Dimension the radial offset between the two circles to 20mm.	 <p>The diagram shows a mechanical part with a central horizontal section of length 300 and a vertical section of length 250. The vertical section has a top diameter of $\phi 50$ and a bottom diameter of $\phi 120$. The horizontal section has a diameter of $\phi 120$. A fillet with a radius of 30mm is applied at the corner where the two sections meet. The angle between the vertical section and the horizontal section is 135°. A red arrow points to the radial offset between the two circles, which is dimensioned as 20mm. A red box in the top right corner contains the text 'Smart Dimension'.</p>
16	Use sketch fillet, R = 30mm, at each corner.	 <p>The diagram shows the same mechanical part as in the previous step, but with fillets applied at the corners. The fillets have a radius of 30mm. The dimensions and geometry are the same as in the previous step.</p>
17	Complete!	 <p>The diagram shows the completed mechanical part. The fillets have been applied, and the dimensions are consistent with the previous steps. The part is now fully defined.</p>

Exercise 1 – Rocker Arm



Exercise 2 – Gasket

