2 – 3D MODELLING

By following this tutorial, you will learn how to:
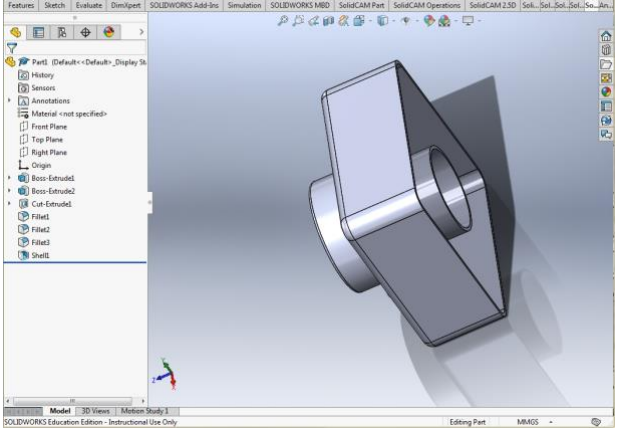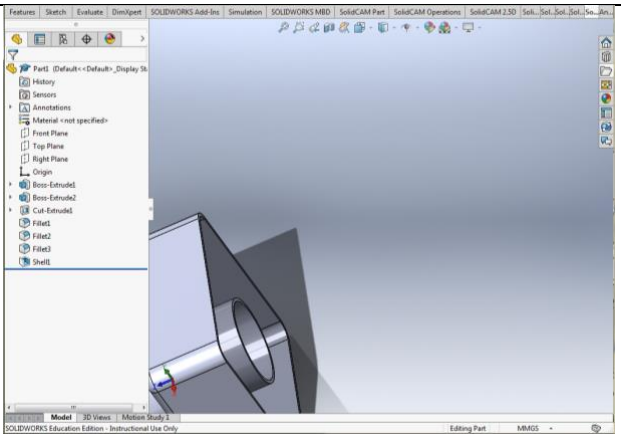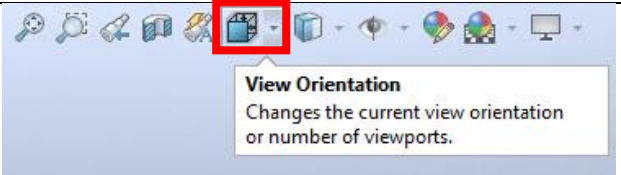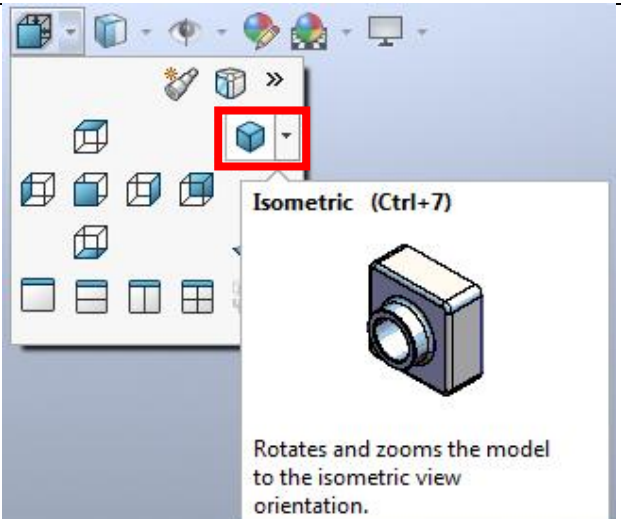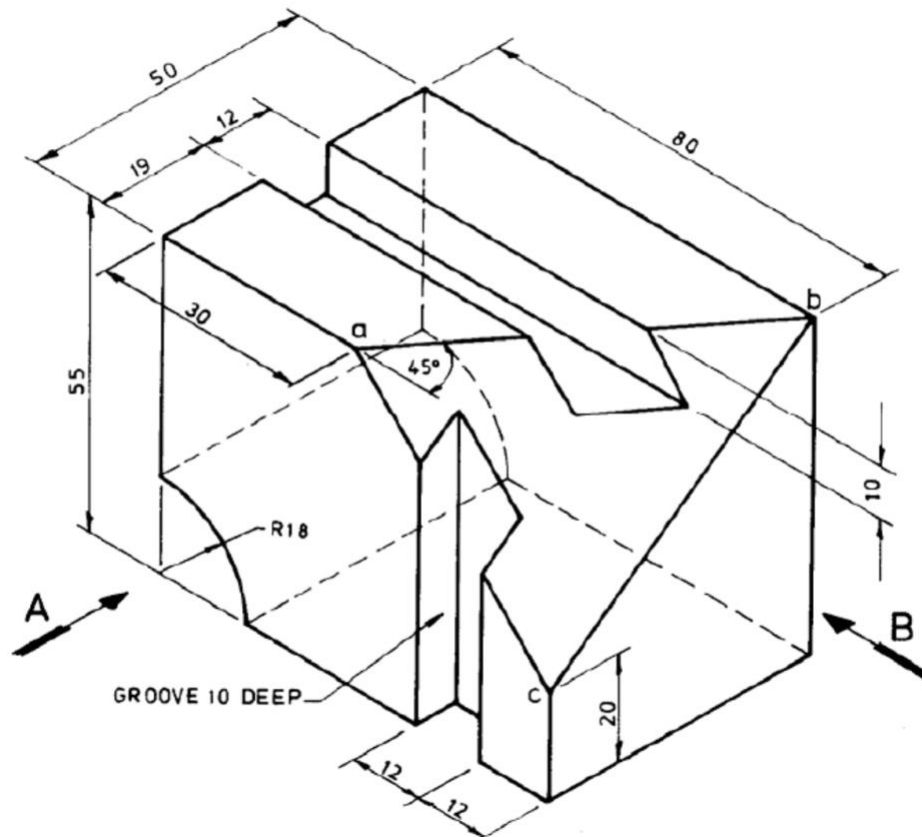1. Navigate around a 3D part file
2. Use View Orientation in SolidWorks
3. Extrude a 2D sketch into a 3D model
4. Create reference geometry
5. Create 3D fillets and chamfers
6. Use revolve

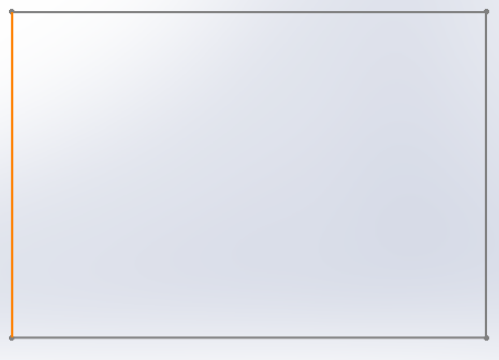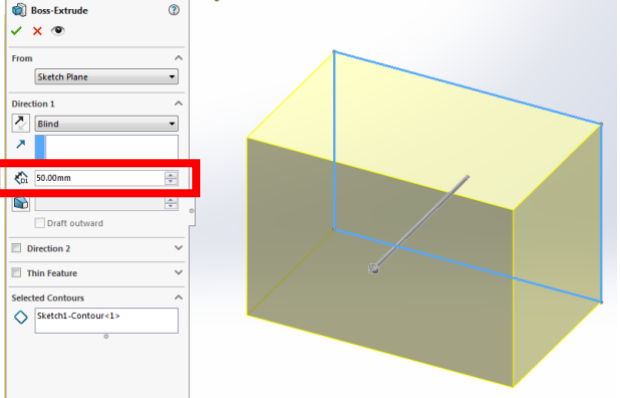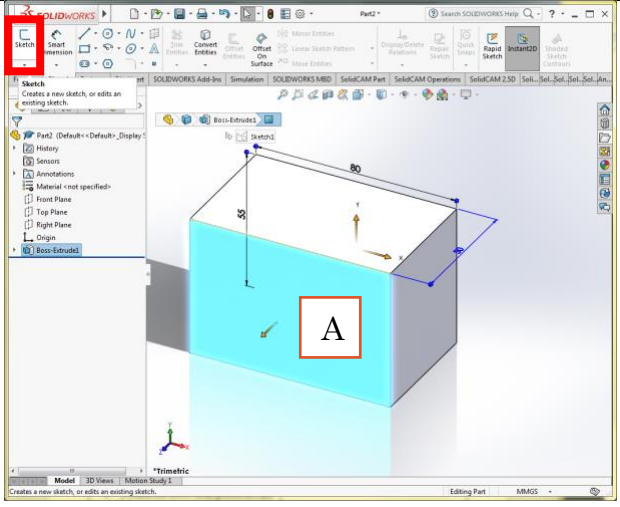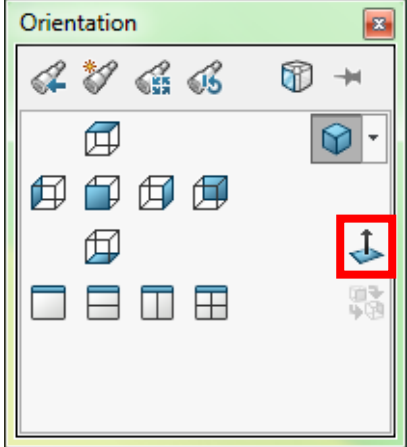| No.	Instruction	Screenshot
	Navigate around a 3D part file	
1	Open an existing part file in SolidWorks. You can use the "2 - 3D Modelling" file provided in Moodle for this step.	
2	Zoom in and out on the part by using the scroll wheel on the mouse.	

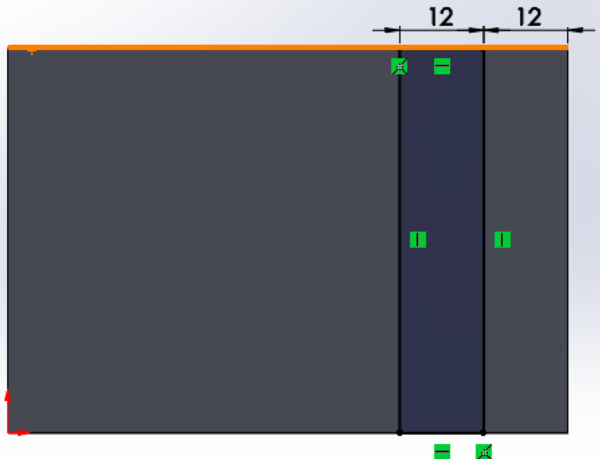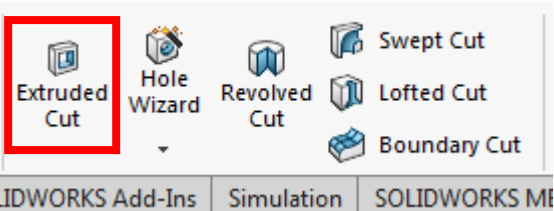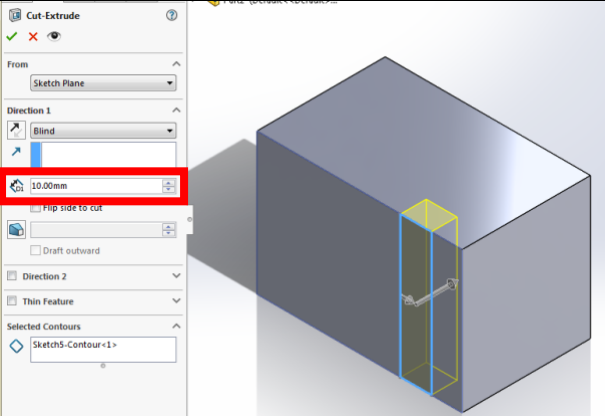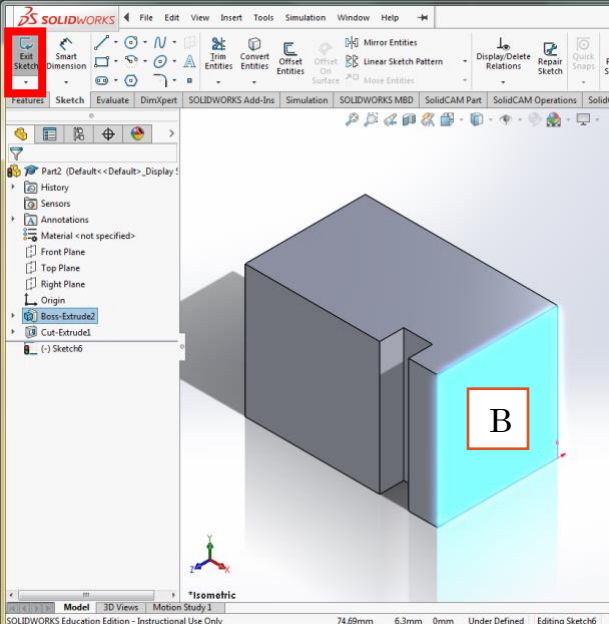| 3	Rotate the part by clicking and holding the scroll wheel on the mouse and moving the mouse.	
4	Move the part by holding down the Ctrl key on the keyboard and holding the scroll wheel on the mouse.	

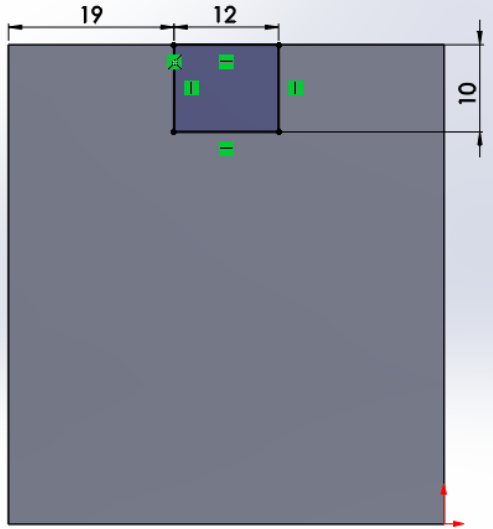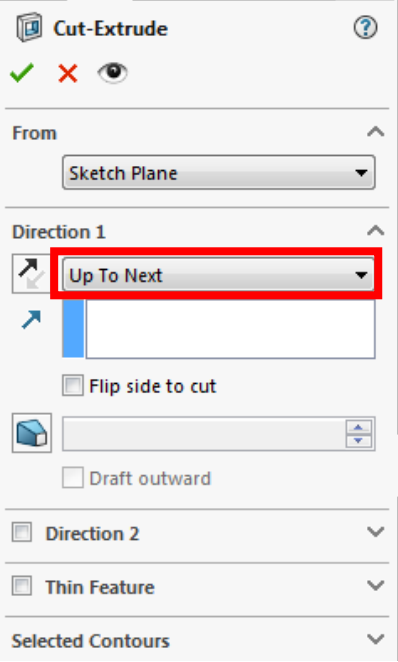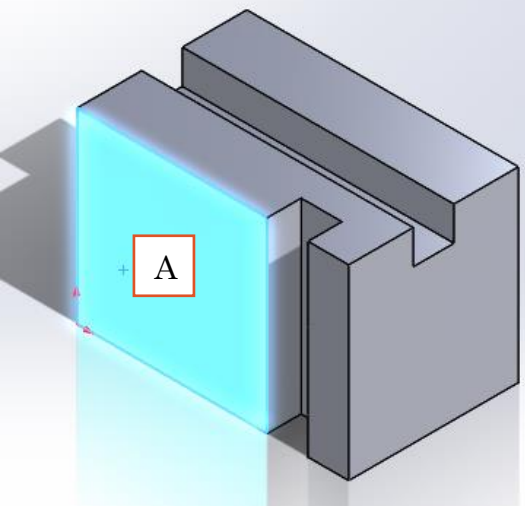| Use View Orientation in SolidWorks		
No.	Instruction	Screenshot
1	Click on View Orientation	
2	Click the Isometric view button to view the part in isometric view.	

To illustrate 3D modelling, the following example will be used.

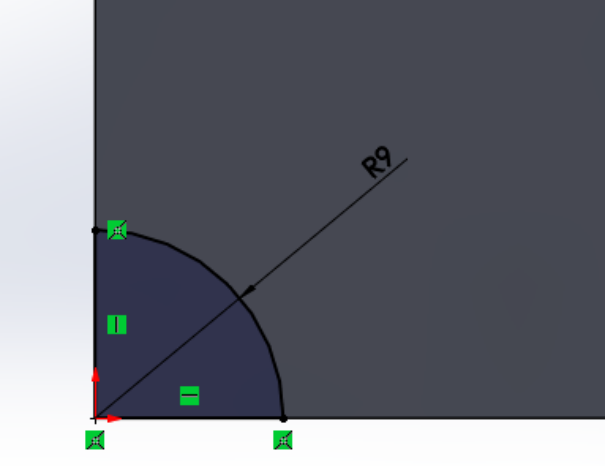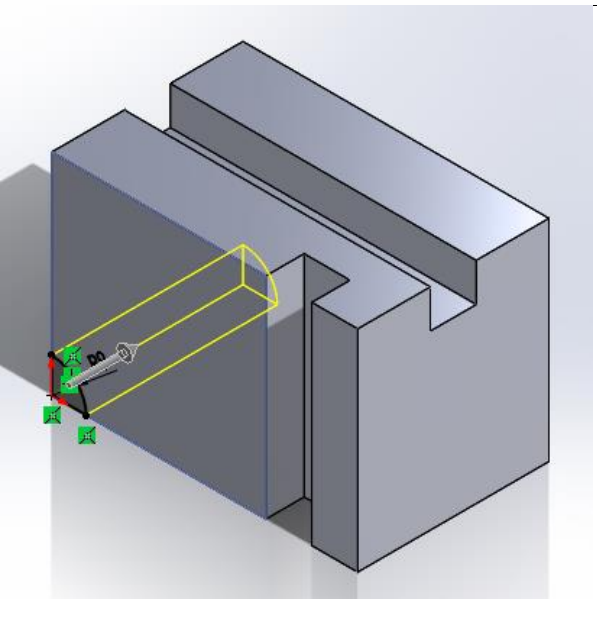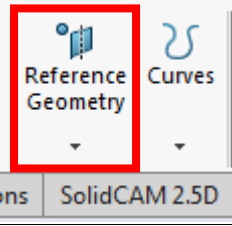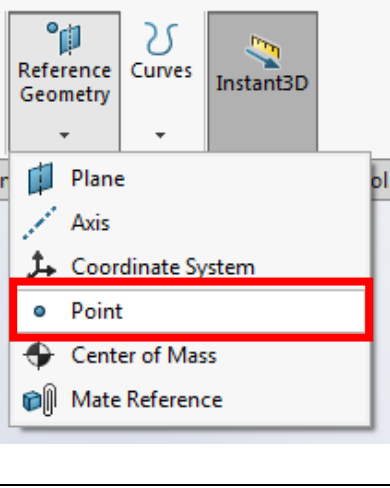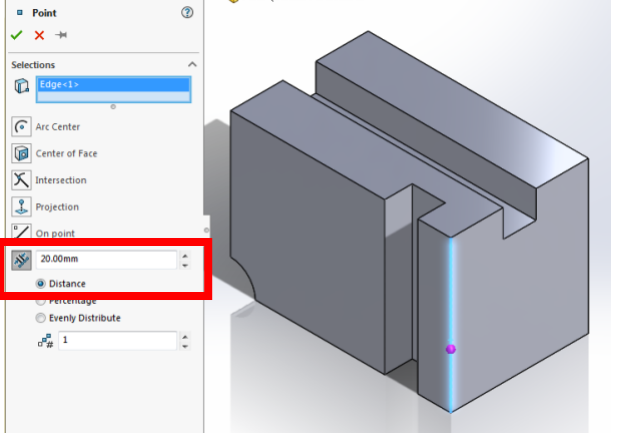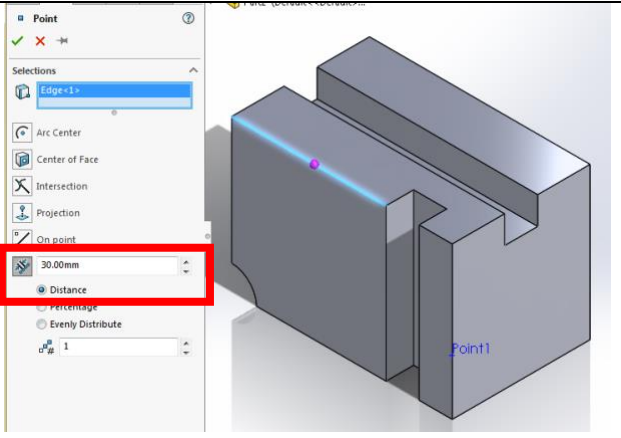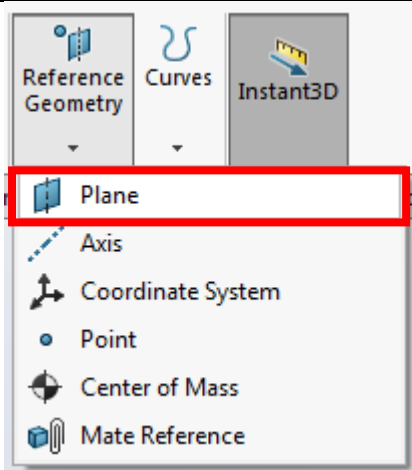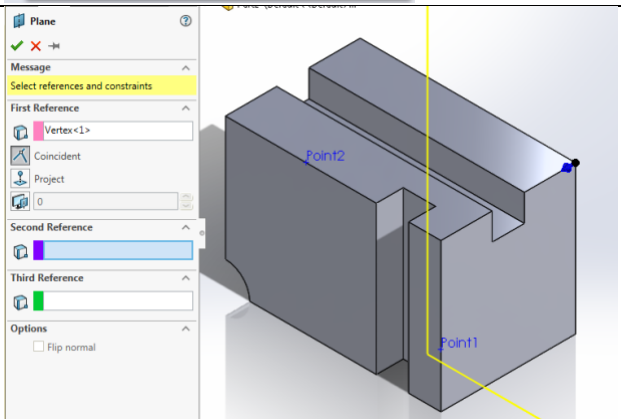| Extrude 2D sketches into a 3D model		
No.	Instruction	Screenshot
1	Create a sketch of the overall geometry of the rectangular box as seen from the perspective of View A. Confirm and exit the sketch workspace.	
2	Go to the Features tab and select the Extruded Boss/Base tool.	
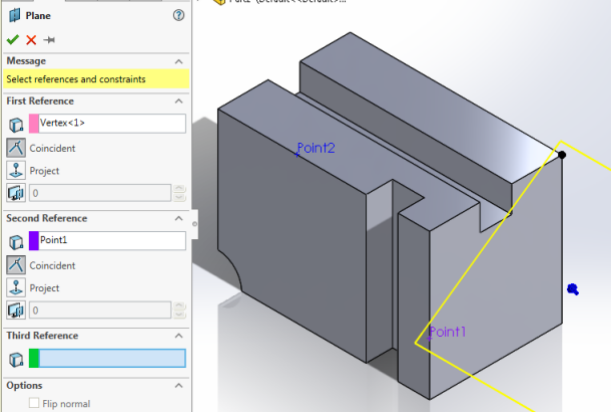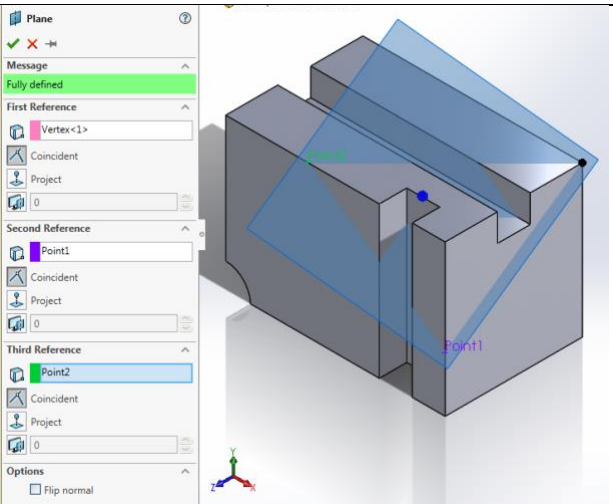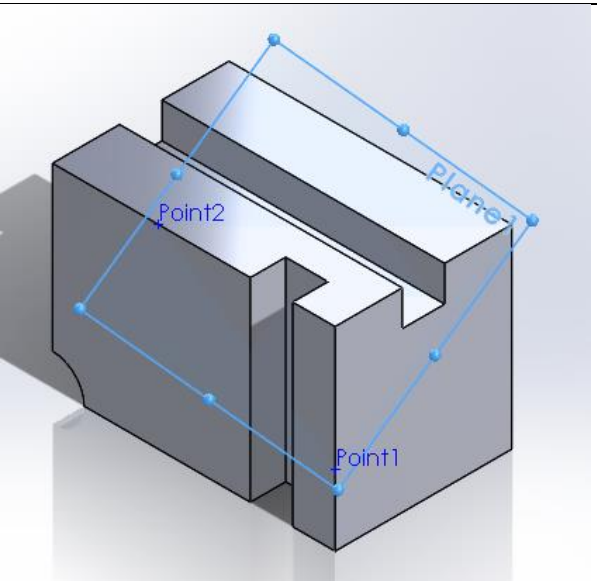
3	Select the existing sketch by clicking on one of the lines.	
4	Enter 50mm for depth under Direction 1 and confirm by clicking the green tick.	
5	Create the 10mm deep groove on Face A. Click the surface representing Face A, and then click the sketch button.	
6	Press the spacebar to bring up the Orientation menu. Click "Normal To".	
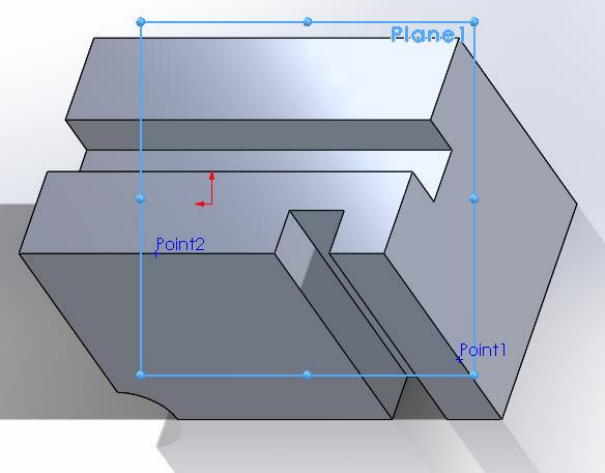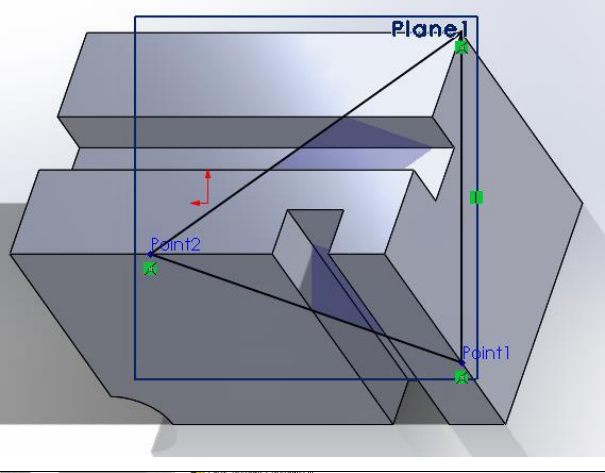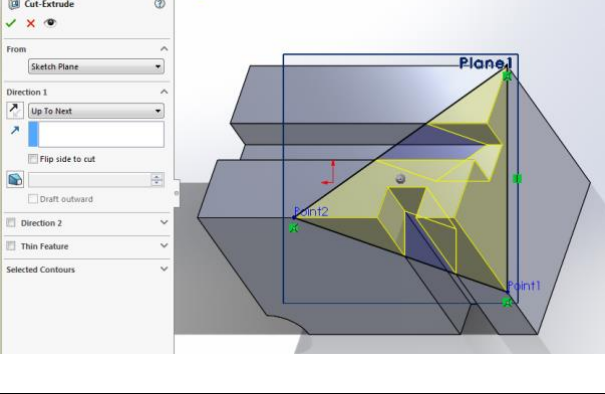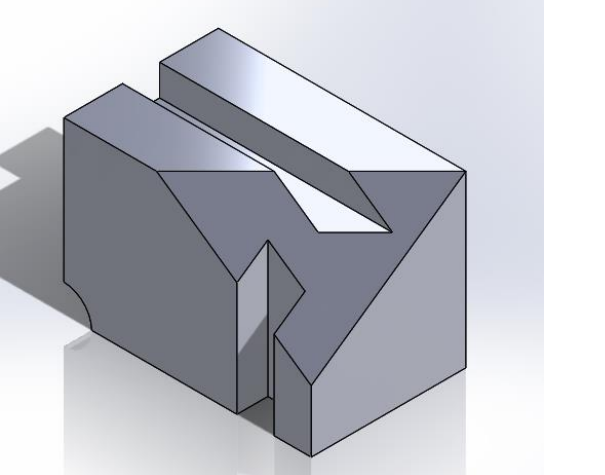
7	Sketch the rectangle 55 x 12mm as shown. Confirm and exit the sketch workspace.	
8	Go to the Features tab and select the Extruded Cut tool.	
9	Enter 10mm for Distance 1 and confirm by clicking the green tick	
10	Cut out the groove on top of the part by selecting Face B. Click Face B of the part and then click the Sketch button.	

11	Sketch the rectangle 12x10, then exit the sketch workspace.	
12	Select the Extruded Cut tool and change the End Condition to "Up to Next".	
13	Cut the circular groove on the bottom corner of the part. Click on Face A, then click the Sketch tool and change the view orientation to "Normal To".	

14	Sketch the corresponding shape on the plane. Radius 9mm.	
15	Select the Extruded Cut tool and use the last sketch. Confirm and exit by clicking on the green tick.	
	Create Reference Geometry	
16	Click the Reference Geometry menu inside the Features tab.	
17	Select Point.	
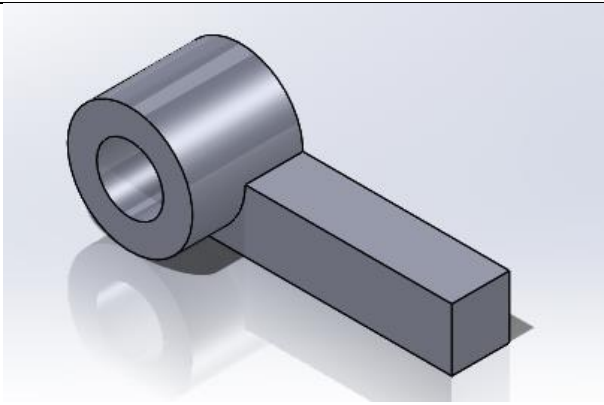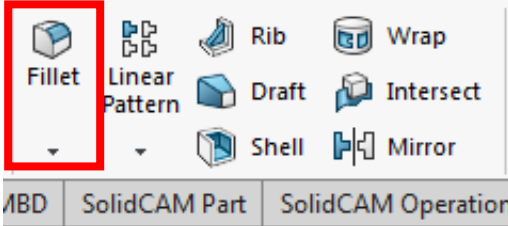
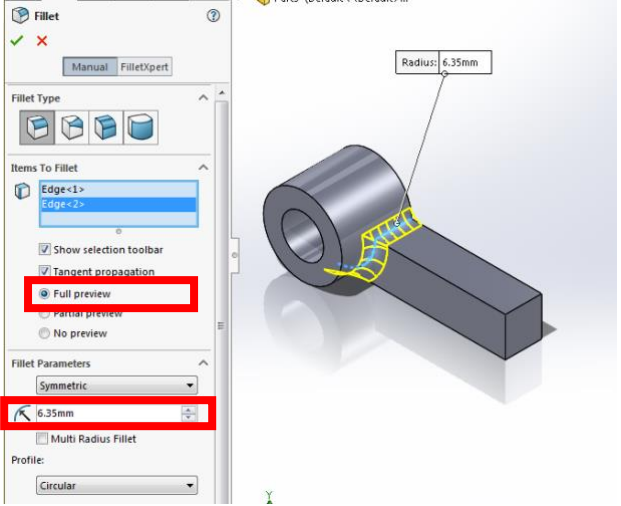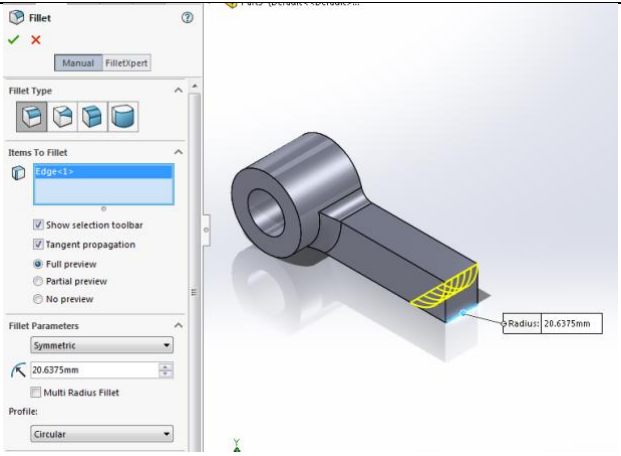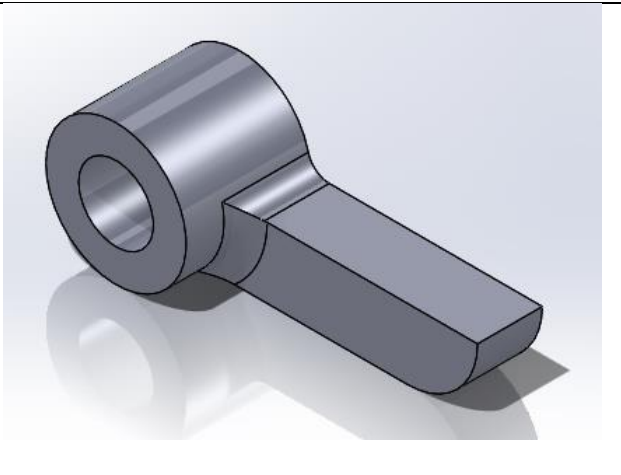| 18	Click the edge shown in the picture. Click the Distance icon in the menu. Enter 20mm as a distance from the bottom of the part. Confirm and exit by clicking the green tick.	
19	Select Point again. Click the edge shown in the picture. Enter 30mm as a distance from the side of the part. Confirm and exit by clicking the green tick.	
20	Select the Plane option under Reference Geometry.	
21	Click the corner as shown to select the First Reference.	

22	Click on one of the two reference geometry points you have just created to select the Second Reference.	
23	Click the remaining reference geometry point to select the Third Reference Click on the green tick to confirm and exit.	
24	Your newly created plane should resemble the one shown.	

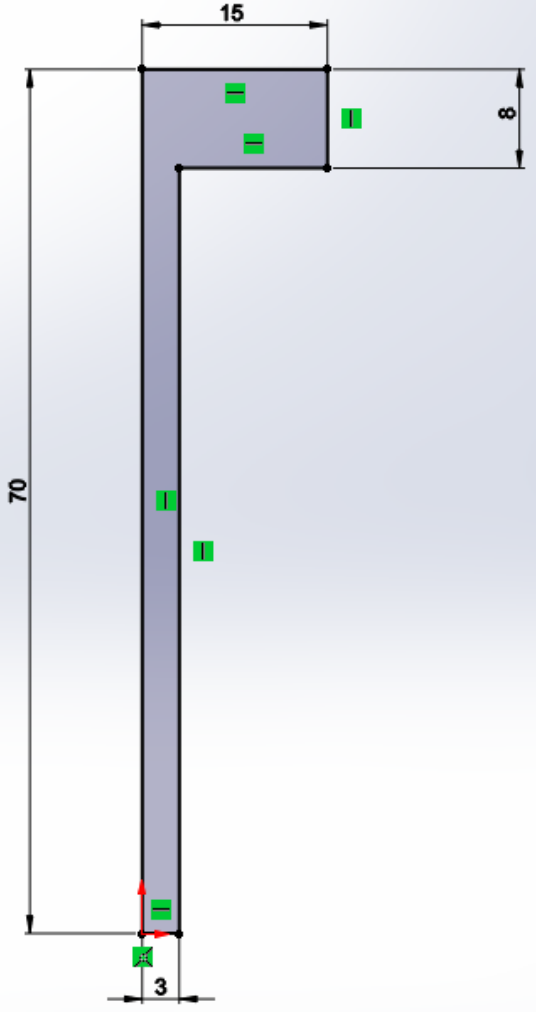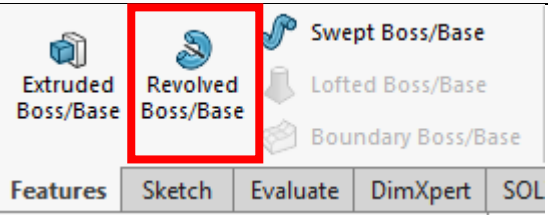| 25	Click the new plane and then click the Sketch button. Change the View Orientation to "Normal To". Note: You can click "Normal To" again to flip the part.	
26	Select the Line tool. Click each of the three points as shown. Confirm and exit the sketch workspace.	
27	Click the Extruded Cut tool under the features tab. Select the last sketch and change the Direction 1 option to "Up to Next". Ensure that the direction is correct. Confirm and exit by clicking on the green tick.	
28	You have now completed the part!	

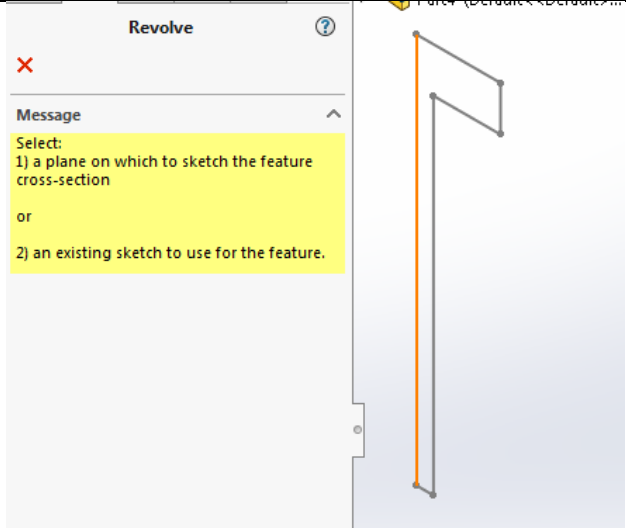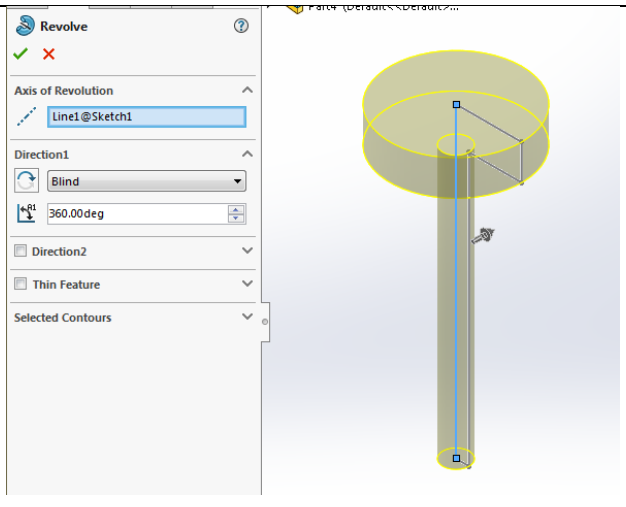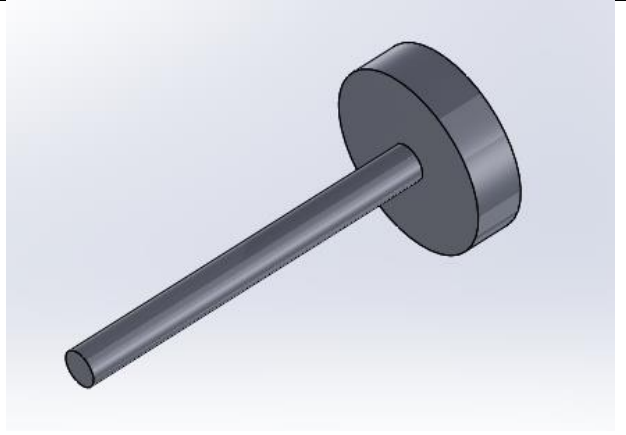For the purpose of illustrating the creation of 3D fillets, the following part will be used:

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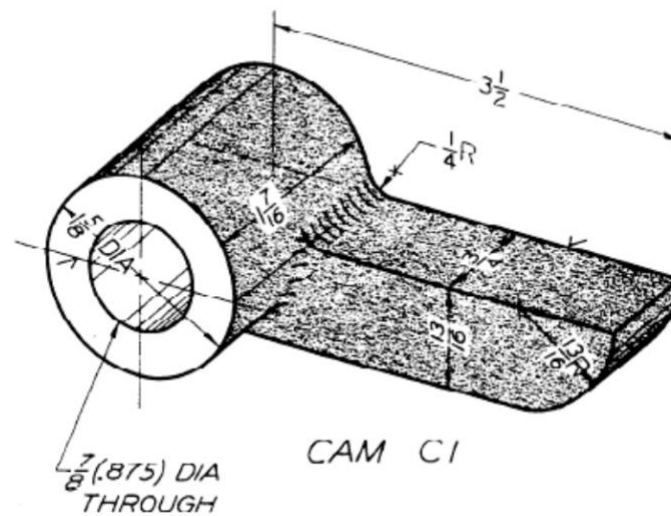
Create 3D Fillets		
No.	Instruction	Screenshot
1	Create the model as shown, excluding fillets.	
2	Click the Fillet button in the Features tab.	

3	Click the edges with a constant size fillet radius. Enter 1/4 in. as the fillet radius. (SolidWorks will convert this to mm) Click the "Full Preview" option. Click the green tick to confirm and exit.	
4	Click the Fillet button. Click the edge at the end of the bar. Enter 13/16 in. as the fillet radius. Click the green tick to confirm and exit.	
5	You have now completed the part!	

Note that the Chamfer tool is under the submenu by clicking on the small downwards arrow under Fillet. The Chamfer tool works in the same manner as the Fillet tool.

Use Revolve		
No.	Instruction	Screenshot
1	Click the Sketch button and select the FRONT sketch plane. Sketch the profile shown in the picture. Confirm and exit the sketch plane.	
2	Click the Revolved Boss/Bass button in the Features tab.	

3	Click the line shown in the picture.	
4	Click the green tick to confirm and exit.	
5	You have now completed the part!	

Note that the Revolved Cut tool in the Features tab works in the same manner as the Revolved Base tool.

Exercises:

BELL CRANK
C I

BEARING CAP
CAST IRON
SI

SADDLE
CAST IRON
SI

LIFTING BLOCK
FILLETS & ROUNDS = R6
CAST IRON
SI

38 DIA

32R

41

32

41

20

38

20

96

12

STOP PLATE
CRS

SI

12

12

40

32R

90

12

29

20

48

10 R

20 R25

20

10

28R

32 DIA

SI

SPACER
CRS