## Episode 2: Digital Modelling

In the last session, we examined how product engineers use Product Lifecycle Management (PLM) software to manage the process of product engineering. In this session, we will look more closely at Siemens NX, used by product designers and engineers to design aeroplanes, smartphones and many household products using digital models.

1. Describing Models

Before we start, its useful to understand the language that designers and engineers use to describe the features and shapes of their models.

Match each letter to the word that describes it, then the definition for each word.

You can use the Siemens NX: Designing The Future Interactive to help find out the answers
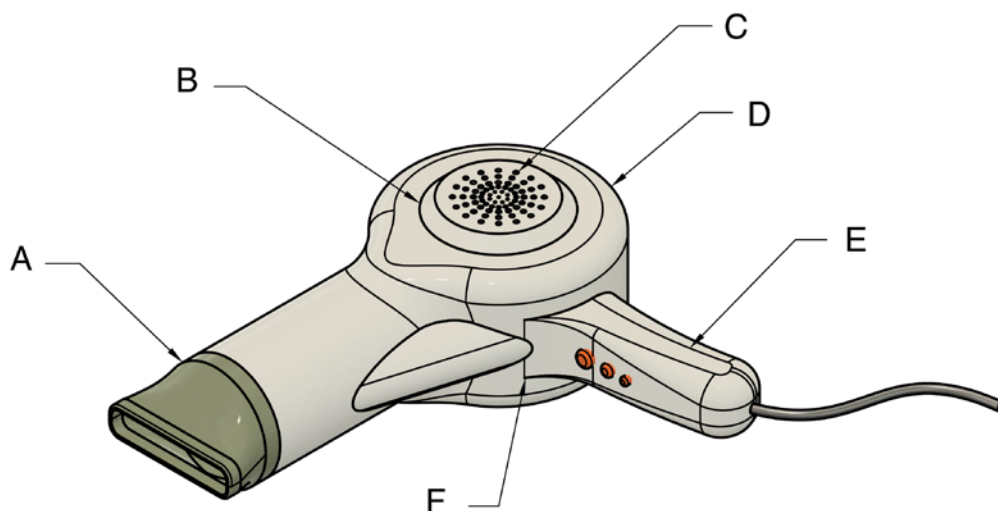
| A		
	VERTEX	A corner
	EDGE BLEND	A circular opening
	PART	A flatly angled edge
	CHAMFER	The flat surface of a part or body
	HOLE	An aspect of a form such as a hole, chamfer or fillet
	FEATURE	A rounded or radiused edge, sometimes called a fillet
	FACE	**A single component of an assembly or product**

2. NX Challenge: Roblet the NX Robot

Having completed the NX challenge from the last session, and having learned some new terminology its time try something more challenging.

Follow the steps below to model this cute NX robot, Roblet. In the next session you will learn how to 3D print your own Roblet figurine!
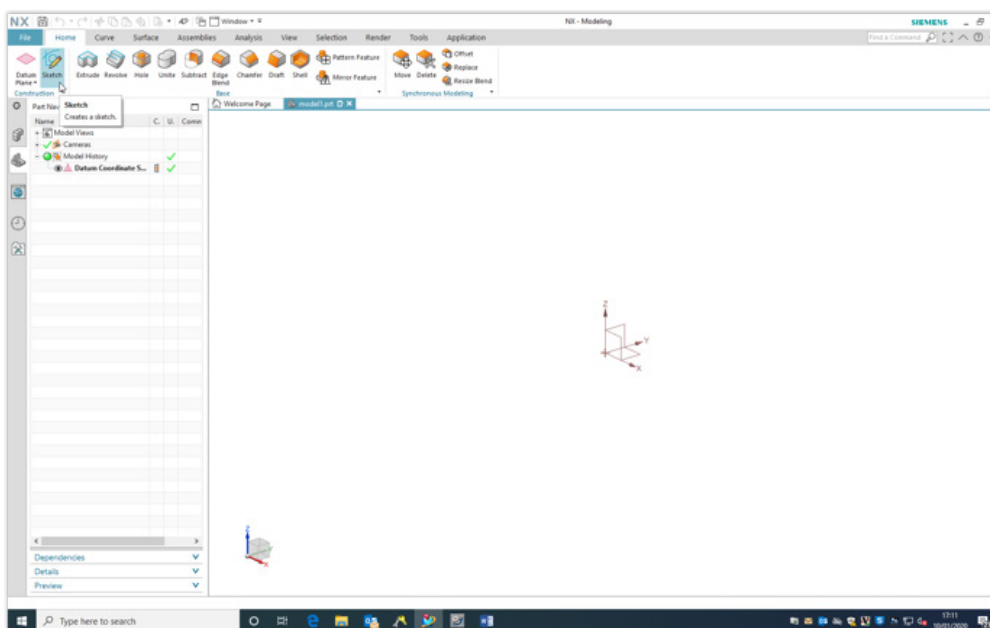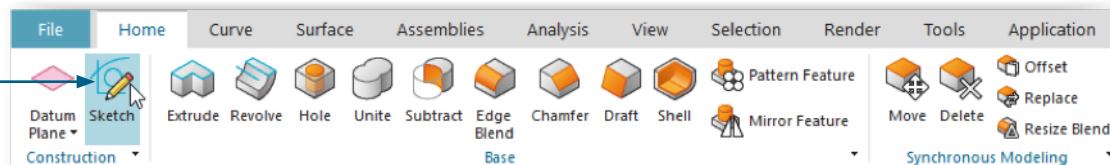
1. Open Siemens NX and click 'New' to start a new project.
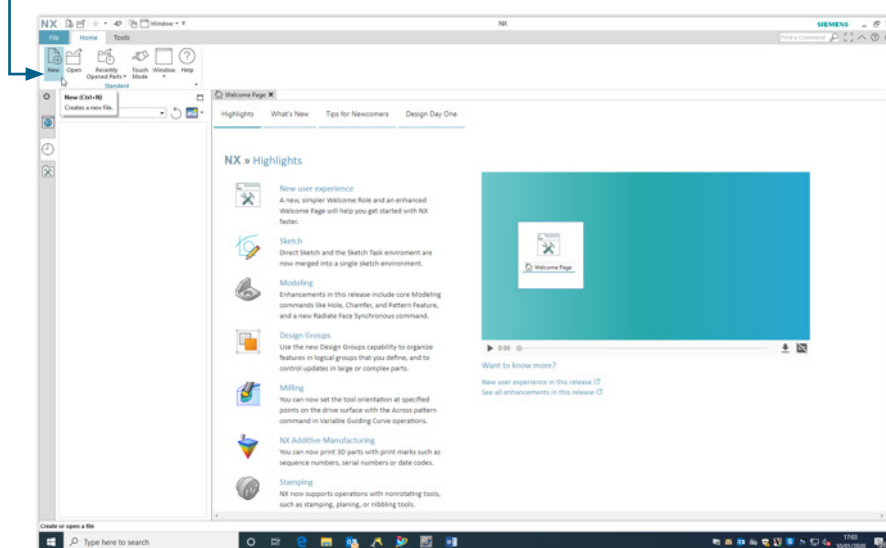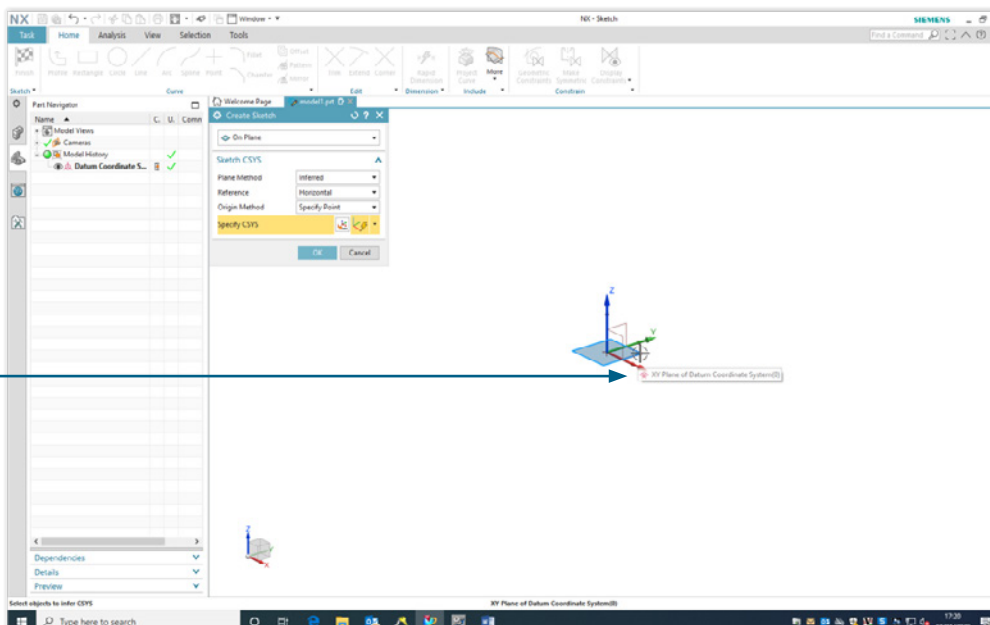

2. Select 'Model' and enter a file name. Don't forget to save this somewhere that you can find it again later.
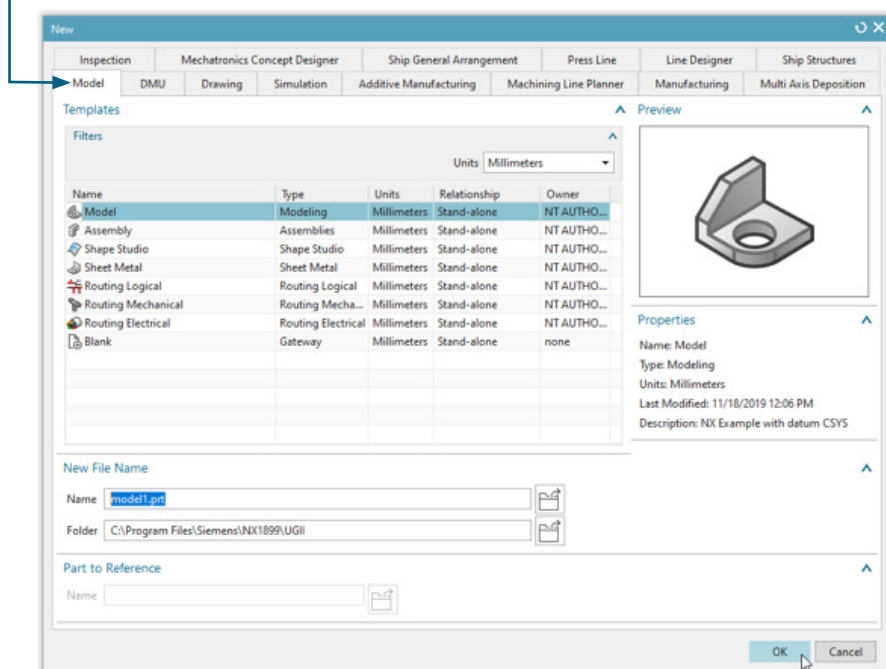
3. Select 'Sketch' from the Base Menu to begin a sketch.

4. Choose which of the three 'planes' you wish to sketch onto. Choose the XY Plane. Once selected, press 'OK' in the Create Sketch Dialog Box

5. Select the Rectangle tool in the sketch menu. Set the Input mode to 'Parameter Mode' and set the width to 25mm and the height to 25mm. Press enter. Use the curser to place the corner of your 25mm square on the sketch origin. Set the angle to 0°. Exit the Rectangle Tool.
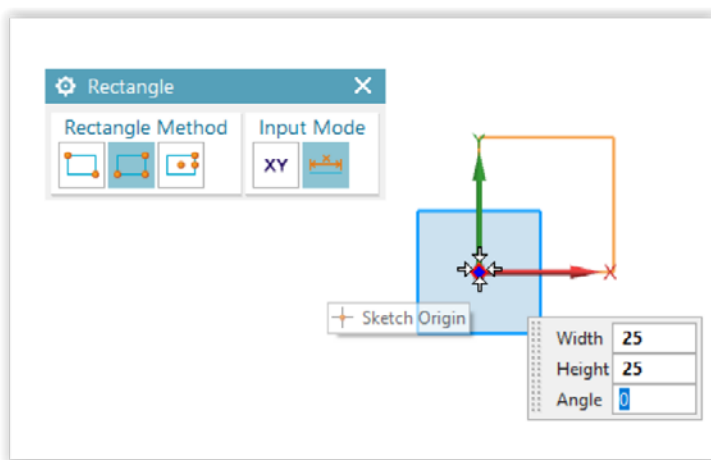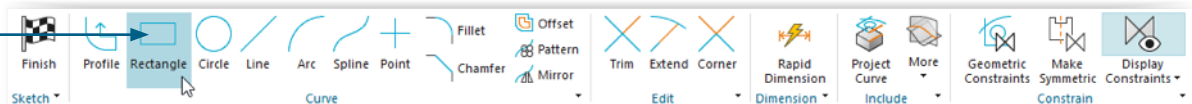

6. Finish the sketch. Name the sketch 'Robot base'. Adjust the view by selecting the View menu, then Fit to Window. You can use your mouse roller to zoom in or out, or alternatively, select view options by right clicking on your work area.
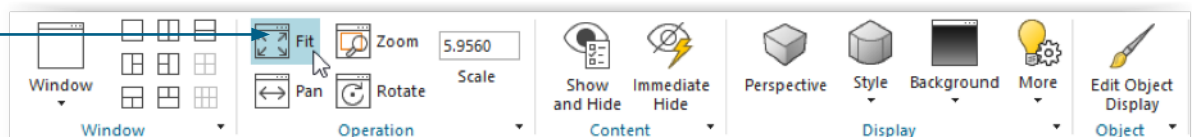

7. Click on your square and select the 'Extrude' tool from the Construct menu. Set the End Distance to 40mm in the extrude dialog box. Select 'Apply' and then Close the dialog box.
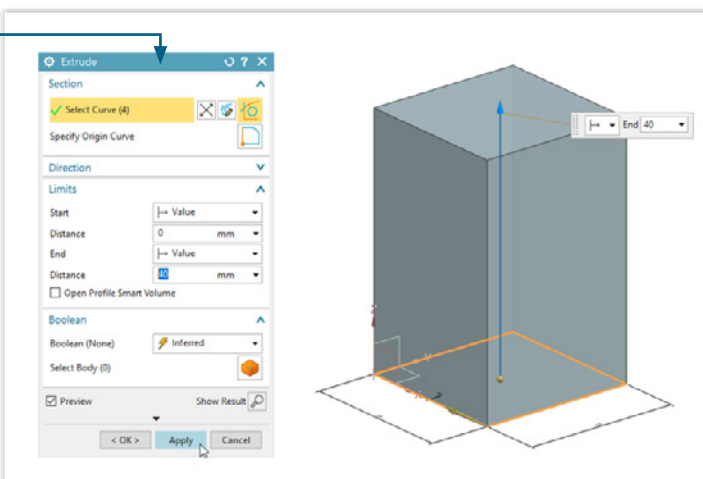
8. Create a new sketch on the front of the cuboid by selecting the 'Sketch' tool and clicking on the front face of the cuboid. Click ok.

9. Sketch a reference line by selecting the line tool. Run your cursor along the top edge of the face until it snaps to the centre, click and drag your line downwards, following the centre line for 14mm. Right click on the line an select 'Convert to Reference'. Reference sketches are ignored by extrusions.
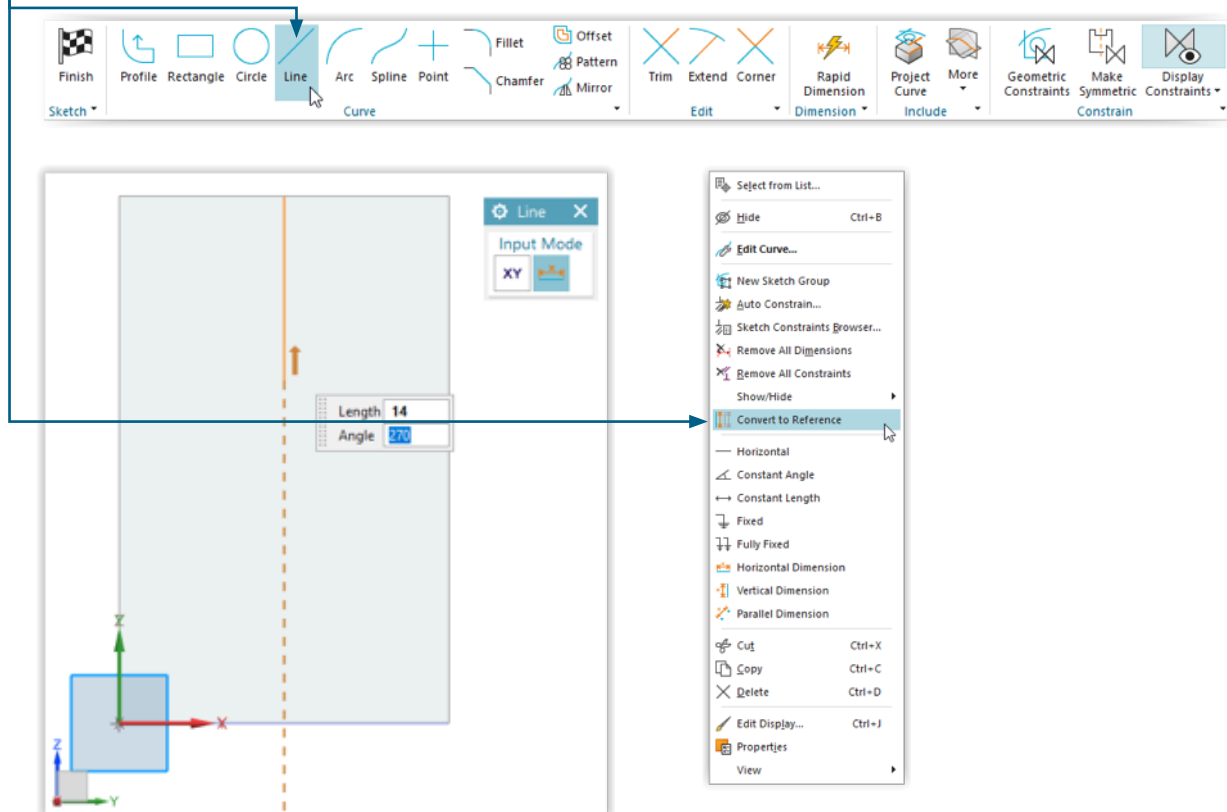

10. Use the circle tool to create an 18mm diameter circle using the lower end of the construction line as the centre.
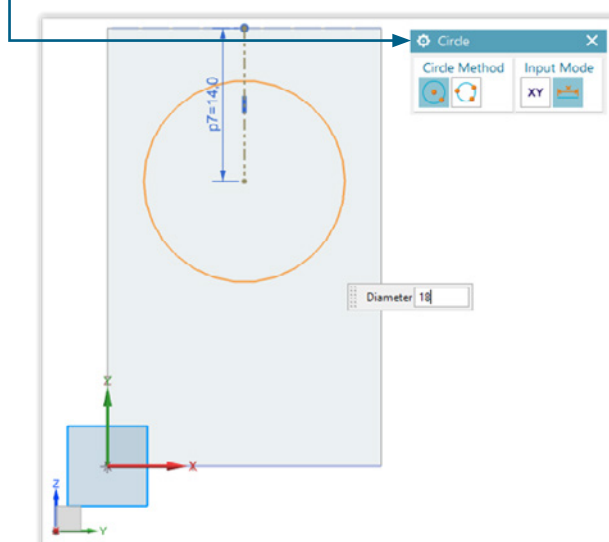
11. Sketch another reference line by selecting the line tool. Run your cursor along the bottom edge of the face until your it snaps to the centre, then click and drag your line upwards, following the centre line for 12mm. Right click on the line an select 'Convert to Reference'.
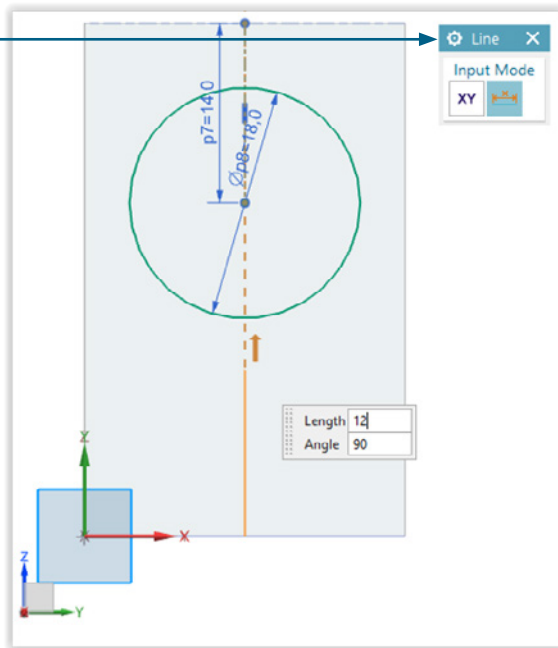

12. Select the rectangle tool. Open sketch a rectangle using the 'From Centre' rectangle method. Use the reference line from Step 12 as the centre. Finish the sketch and then name it 'screen and controls'
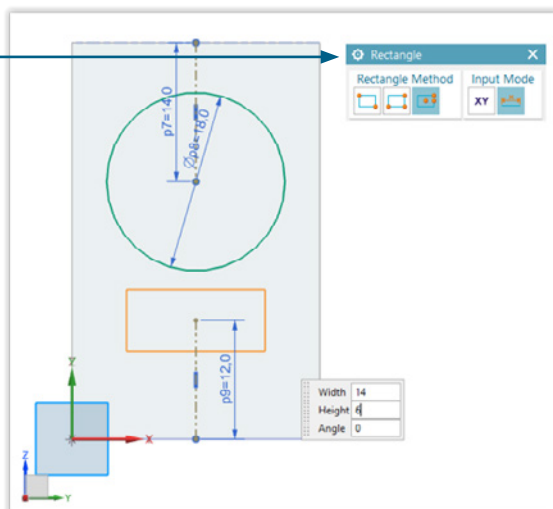
13. In the part navigator, change the Camera view to "Isometric".
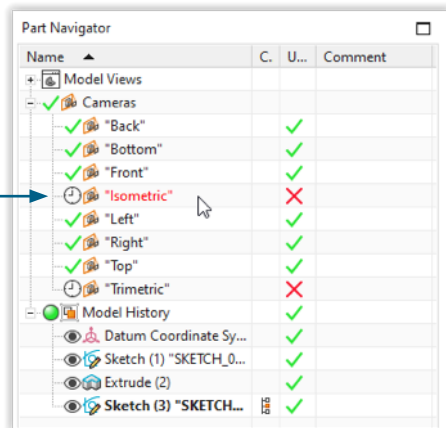
14. Select 'Edge Blend' from the Base menu. Select the left and right, top and bottom edges of the cuboid and set the radii to 10mm. This will fillet or 'round off' the selected edges.
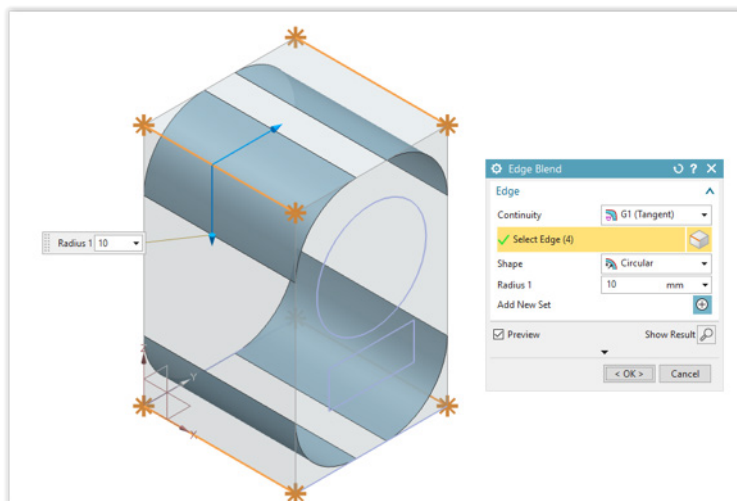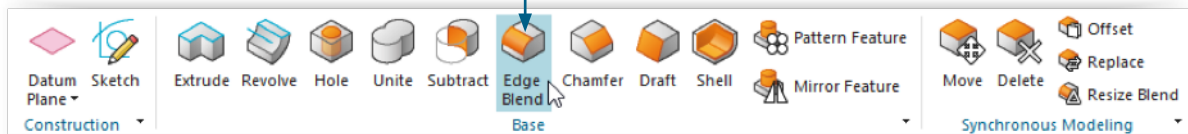
15. Select the sketch tool and create a sketch on the left face of the robot. Draw the shoulder using a 10mm diameter circle in the centre of the sketch. Add a 15mm reference line downwards from the circle centre. Use this as a reference to sketch a 15x5mm rectangle using the centre method, forming the arm.
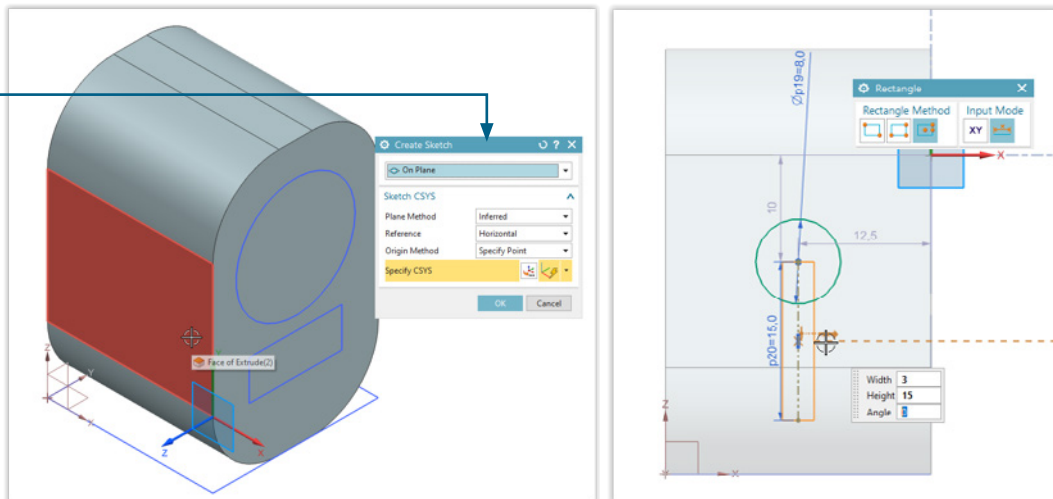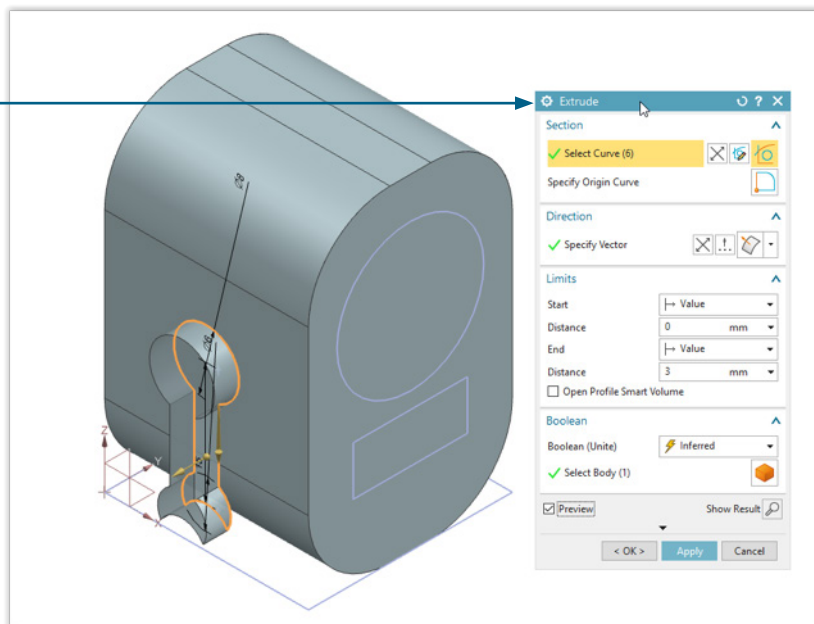

16. Add two 6mm diameter circles to the end of the arm.
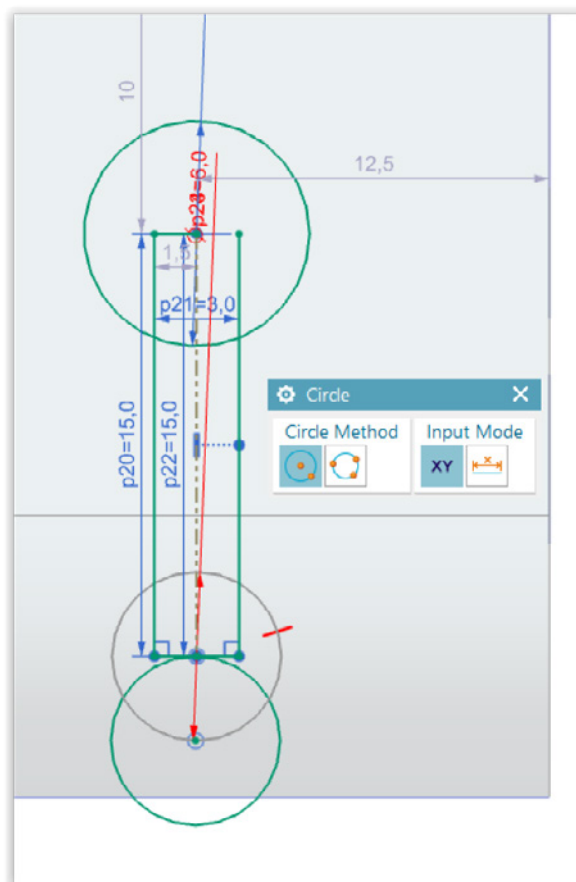
17. Use the 'Trim' tool to remove any extra lines until you are left with one shape as shown below. Finish the sketch.
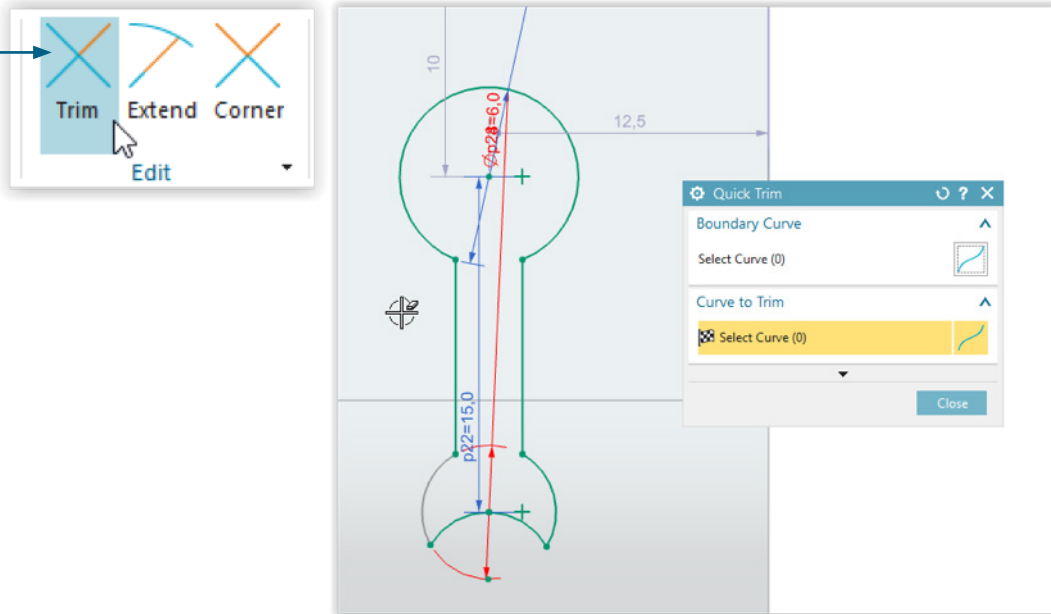

18. Extrude the arm 3mm.

19. We will now mirror the arm on the other side of the robot's body. Create a new plane by selecting 'Datum Plane' and selecting the left face of the robot body. Offset the plane from the surface at a distance of 12.5mm. You may have to reverse the direction to get the plane to appear along the centre of the robot.
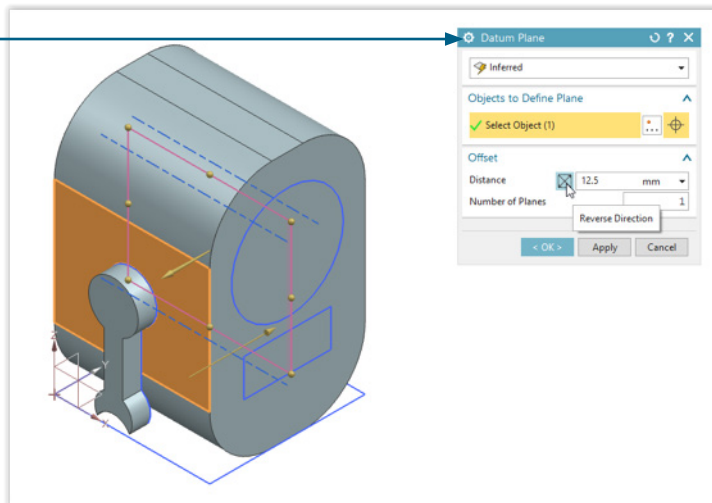

20. 20 Select 'Mirror Feature' from the Base menu. Select the arm as the feature to be mirrored and the plane from Step 19 as the Mirror Plane. Select 'Ok'. The arm should now be mirrored onto the other side of the robot.
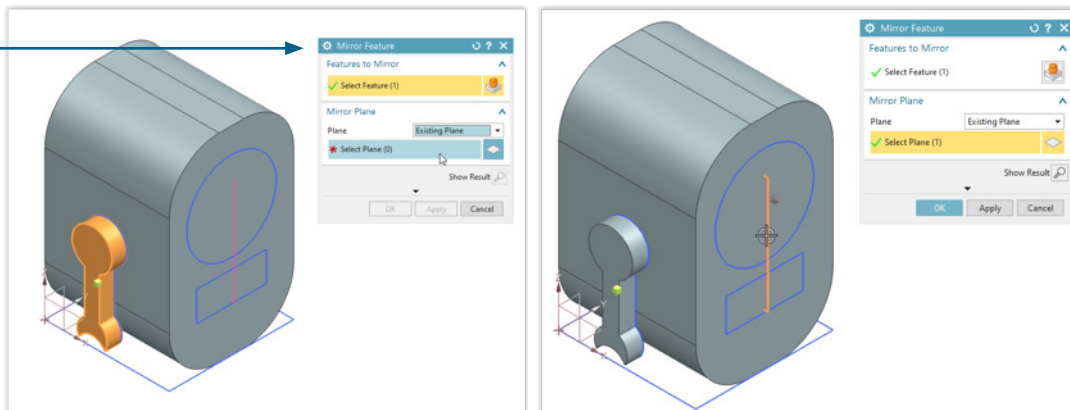

\21. You will now model the robot's wheel. Create a new sketch on the underside of the robot. You may have to rotate your model using the view menu functions. Sketch a 2x2mm square in the centre of the plane. Finish the sketch. Extrude the square a distance 5mm
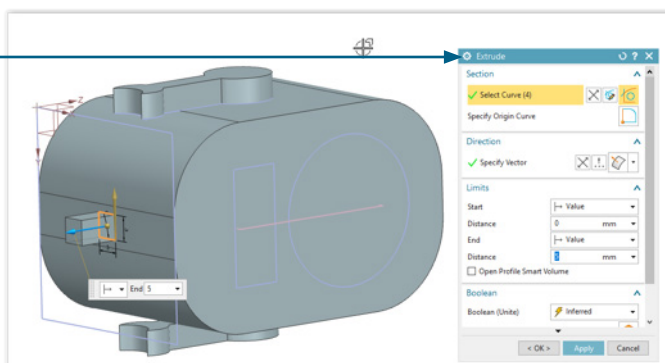
22. Create a new sketch on the left side of the cuboid created in step 21. Sketch an 8mm diameter circle as shown. Finish the sketch and extrude the circle 3mm. You may have to reverse the direction of the extrusion.
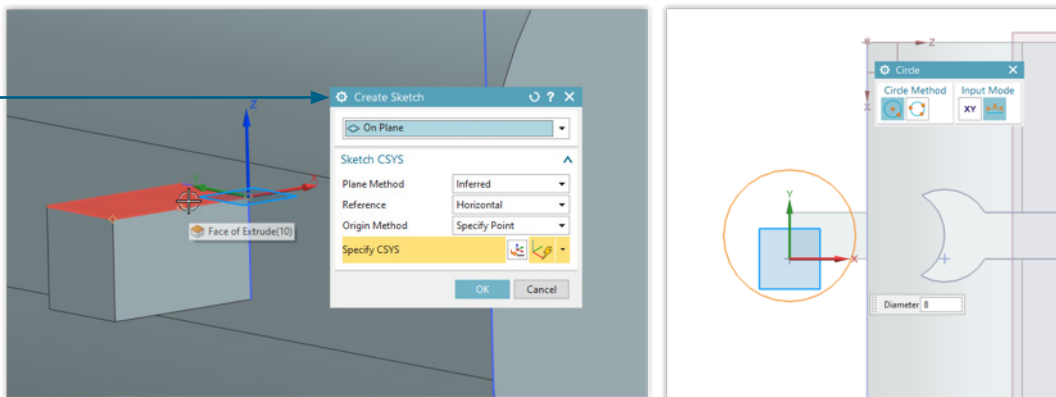

23. Add a 3.5mm radius Edge Blend to the edges of the robot's body.
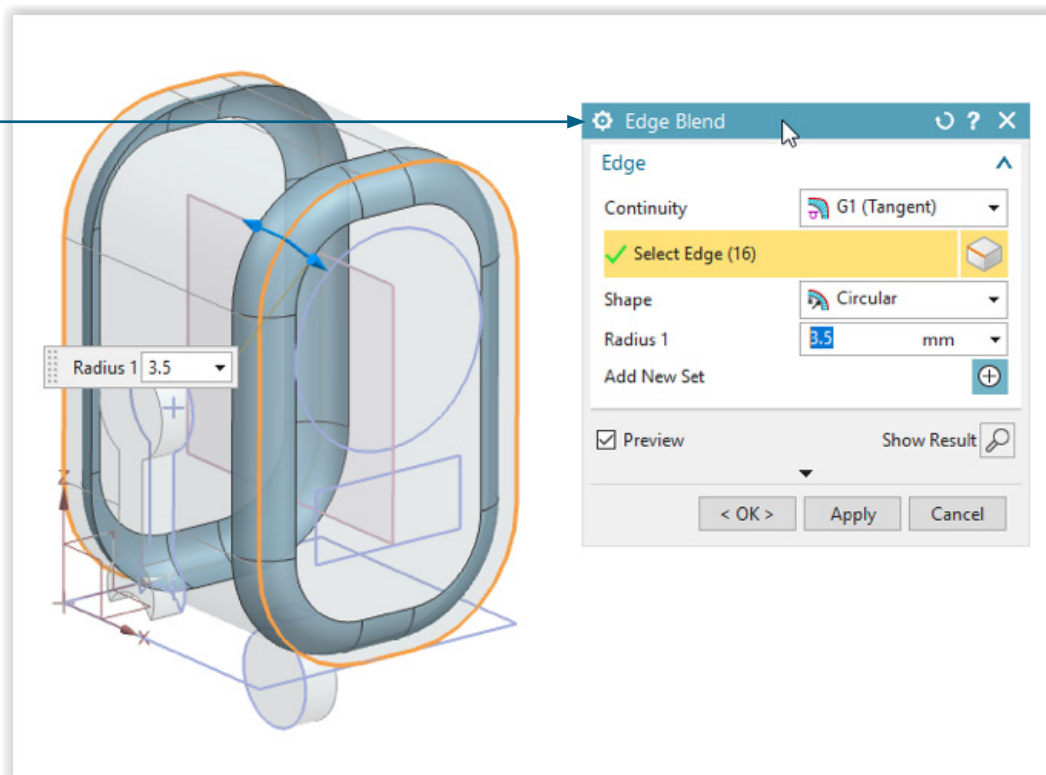
24. Extrude the robot's face screen and control panel 1mm. Add a face and controls, using your own sketches and extrusions. You may wish to use the face design from your Emotion badge, or experiment with the Sketch menu tools.
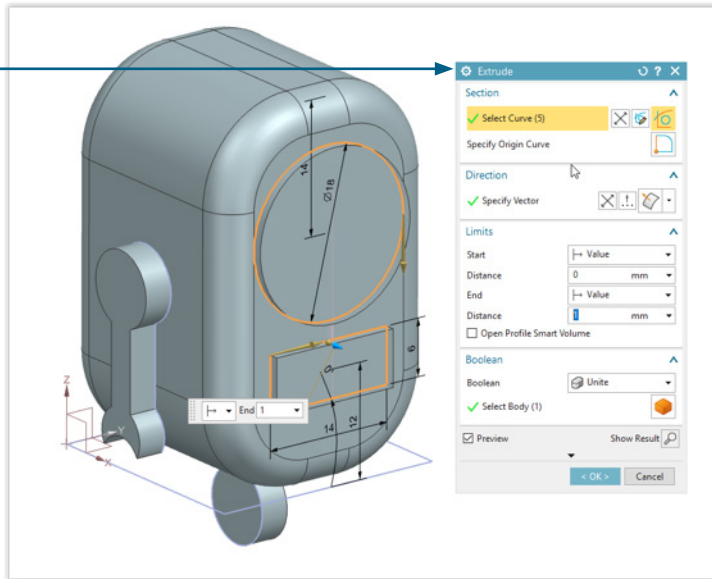
25. Using the Chamfer tool, add a 0.5mm chamfer to the robot's face screen. A chamfer is an angled edge.
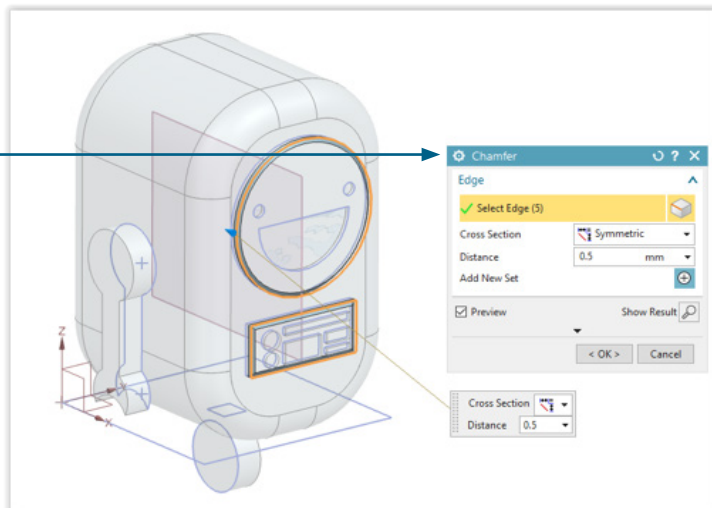
26. Your robot is now complete. You can adjust the lighting and background in the Display menu of the View tab.
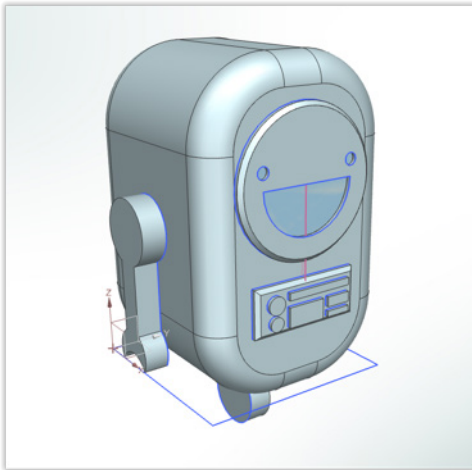
NX Extension Challenge: Customise Your Roblet

Try out different facial expressions, or use your new NX skills to customise your Roblet, with an antenna, a hat, or other accessories. Roblet the NX Robot loves to accessorise!

For further NX tutorials, the Siemens NX Education website **[Link]**