Creo Parametric: **Creation of a simple part and flat pattern in the Sheet Metal module (1)**

Level: basic, Year 10/11
Contribution to D&T, Maths, Science: design and creation of nets for construction of containers, nets of geometrical shapes.

This method of working with sheet metal is to **create a solid** first then shell it before opening the sheet metal module. The alternative is to create the model within sheet metal from a number of flat pieces joined together.

1. Create and name a new **Part**

2. **Extrude** a 200x100 rectangle 50 mm to give a solid and **Shell** from the top surface to 2mm

3. In **Operations** tab click Convert to **Sheetmetal**

4. In **First Wall** tab select **Driving Surface** and click on inside bottom surface and tick OK.
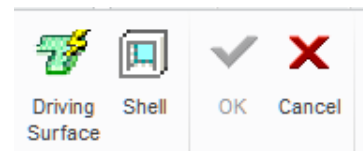
5. Sheetmetal ribbon menu opens.

6. Select **Conversion** there are 4 optional features that can be defined.

7. Select **Edge Rip** hold down Ctrl and click on the 4 vertical edges (they turn green), then **accept.**
   Lower edge bends are applied automatically

8. Select **Corner Reliefs** change from V Notch to **Obround** then **accept**. Conversion is complete.
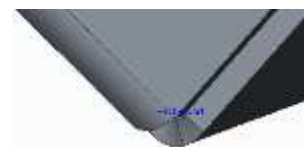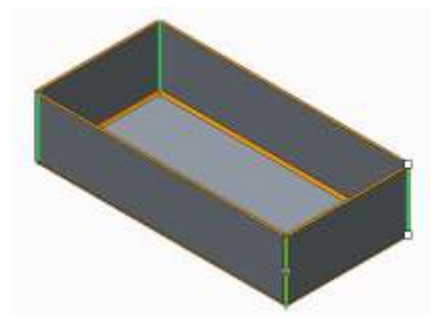
9. Select **Flat Pattern** (bottom surface is automatically selected) and **accept**. The flat pattern that results is labelled with the bend angles and relief at the corners.
   The outline can be exported from the drawing interface.

JM 19/11/12