


Creo Parametric: Creation of a simple part and flat pattern in the Sheet Metal module (2)

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

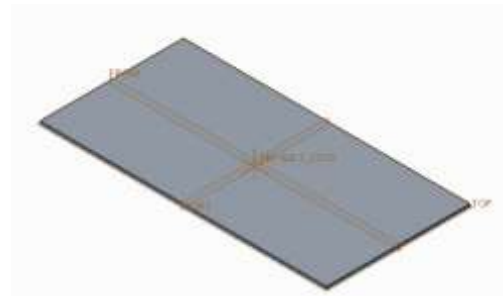
Create and name a new **Sheetmetal** part.

You have the choice of sketching first (external sketch) or selecting the feature and then sketching (internal sketch), as with extrusions.



1. Create a **Planar** wall 
2. Select the TOP plane and add centrelines
3. Sketch a rectangle from and make its size 200 (H) x 100 (V)
4. Click on the ✓ to finish the sketch.

5. Enter **2** (mm) in the top left box (for the thickness) and **Accept**

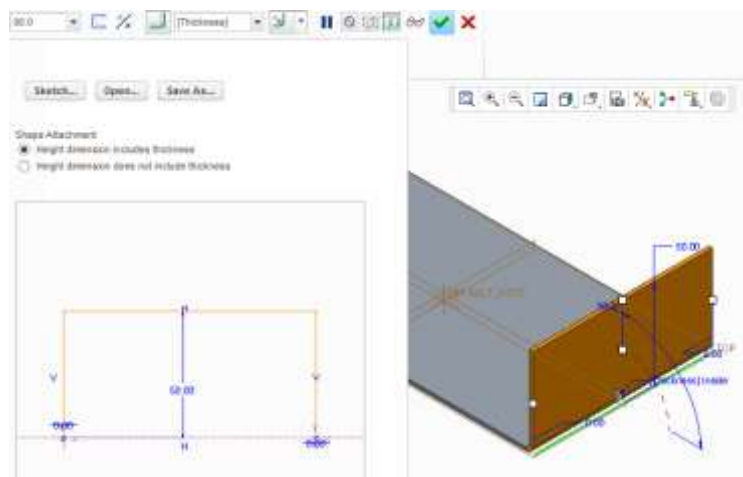


6. Select **Create Flat Wall**  and click on the lower edge of the first wall (zoom in if necessary)

7. The height of the new wall can be dragged to size or a value of **50** entered, then **Accept**.

*Note that you can use the **Shape** menu to sketch a shape for the wall and dimension it, but it must be open at the attachment end.*


8. Repeat this 3 more times to get the other walls in the same way (or you can use Copy and Paste by selecting the first wall and then the attachment edge each time you paste).



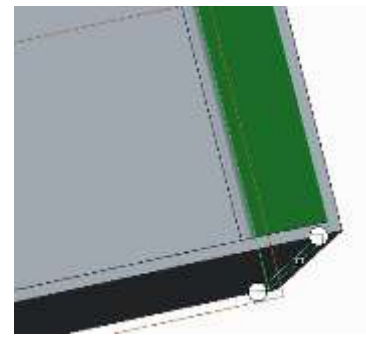
There will be small gaps at the corners!

9. To fill the gaps, first select an **Edge** (hold down the Alt key) and then click **Extend**




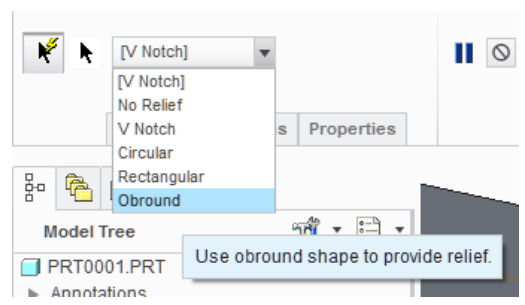
10. Select  **Extend the wall up to the reference plane** then select the inside plane of the adjacent wall and **Accept**


11. Do this for the other gaps



12. Finally the corner reliefs can be defined.

13. Select **Corner Relief**  (Engineering tab) and select **Obround** instead of V Notch. Note all corners are selected by default.



14. Click on  to create the flat pattern