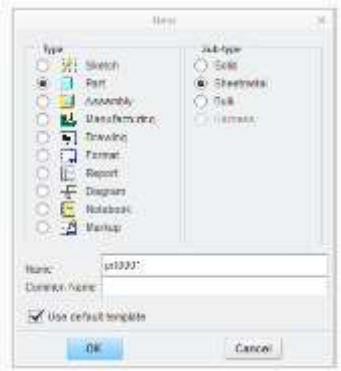
Creo Parametric: **Creation of a simple part and flat pattern in the Sheet Metal module (3)**

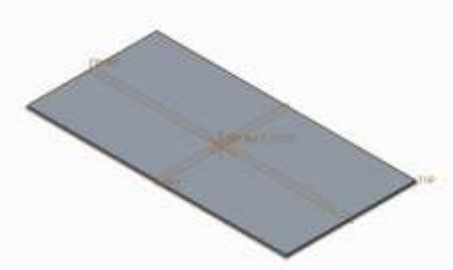
This method of working with sheet metal is to create the model within sheet metal from **a flat piece with a number of flanges**. The shape of the flange cross-section is driven by a sketch.



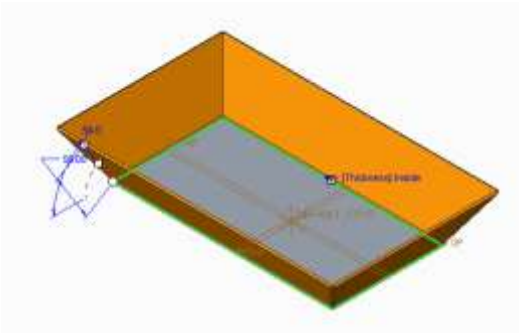
Create and name a new **Sheetmetal** part.

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.

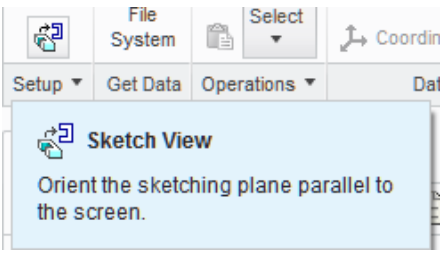


5. Click on the green tick to finish the first wall.

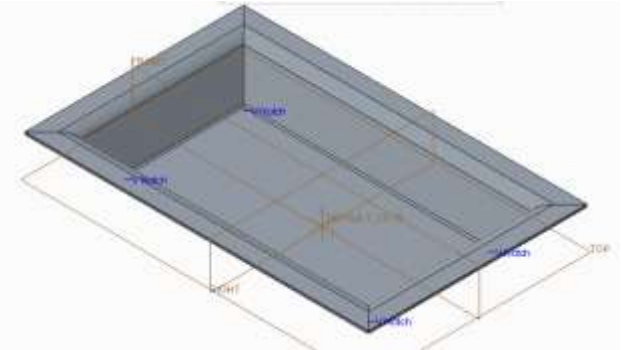


6. Select **Flange w**all  Hold down Shift and click on the four lower edges of the first wall. The edges selected are red lines and four walls will appear:
7. Change the length to **50** and the angle to less than 90º (don't accept yet!)

8. Click on **Shape**, **Sketch** then **Sketch** on the floating dialogue box.
9. Click on Sketch View to orient the view and extend or modify the sketch line to give the required profile of the box



10. Click on the ✓ to finish the sketch.

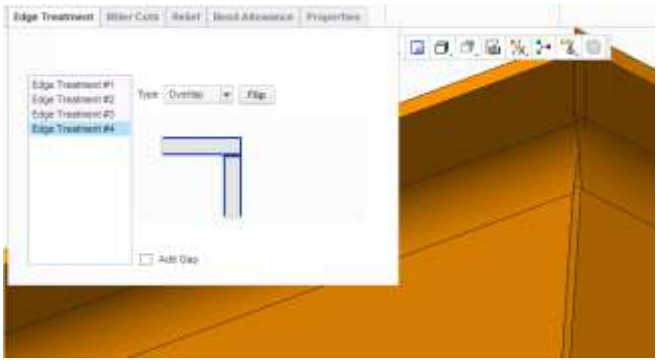


11. Click on the green tick to finish the sheet model

There will be small gaps at the corners.

*These can be removed by clicking on **Edge Treatment tab** and changing each edge to **Overlap**.*

*Right-click Flange1 and **Edit Definition** to go back to do this if you need to.*



12. Select  to create the flat pattern