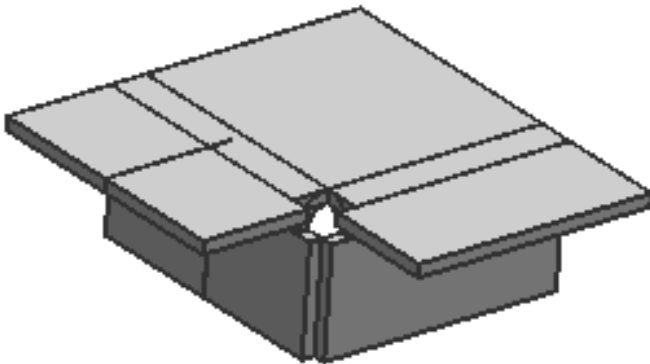




Flat Solid overview

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Flat Solid allows you to create a flat representation of the part in the same file as the formed sheet metal part. The flattened version of the part is associative to the formed version. When you flatten a sheet metal part this way, a **Flat Solid** feature is added to the end of the **Model History** in the **Part Navigator**. If the part contains deformation features, they are retained in their formed condition. If the sheet metal model changes, the flat pattern automatically updates to include the new feature.



- Model History
- ☒ Fixed Datum Plane (0)
 - ☒ Fixed Datum Axis (1)
 - ☒ Fixed Datum Axis (2)
 - ☒ SB Tab (3)
 - ☒ SB Flange (5)
 - ☒ SB Flange (7)
 - ☒ SB Closed Corner (9)
 - ☒ SB Edge Rip (13)
 - ☒ Sketch (17) "SKETCH_..."
 - ☒ SB Flat Solid (18)

Where do I find it?

Application	NX Sheet Metal
Toolbar	NX Sheet Metal→Flat Pattern list  →Flat Solid
Menu	Insert→Flat Pattern→Flat Solid



Create a flat solid

[Overview](#)[How To](#)[Related Topics](#)

1. Click **Flat Solid**.
2. Select a face that you want to orient upward.
3. Select an edge to define the X-axis and origin.
4. (Optional): Select **Move to Absolute CSYS** to move the flattened representation of the part to the model space coordinate system.
5. Click **OK** to finish the flat representation of your part.