

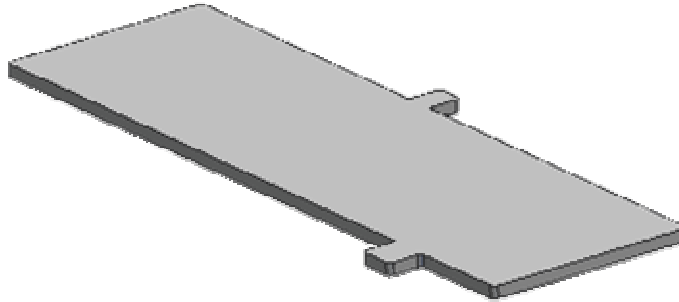
# Creating standard parts such as brackets and simple enclosures

In a typical NX Sheet Metal workflow, you:

1. Set the default values for sheet metal properties. (See [Changing the Defaults](#) for more information.)
2. Sketch the shape of the base feature, or use an existing sketch/section.
3. Construct the base feature that defines the part's shape. This base feature is normally a tab, but it can also be a contour flange or a lofted flange.
4. Add features, such as flanges, jogs, and bends to further define the basic shape of the formed sheet metal part.
5. Apply unbends to flatten the bend areas where needed, and place holes, cutouts, embosses, or louver-type features on the part.
6. Rebend flattened bend areas to complete the part.
7. Create a flat pattern of the part for drafting and later manufacturing.

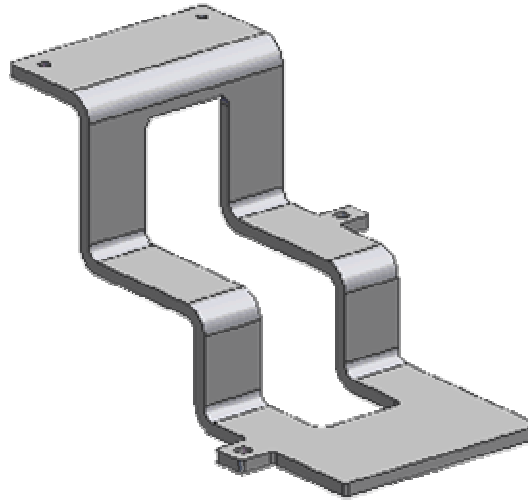
## Constructing a base feature

The [Tab](#) command allows you to construct a flat feature of any shape using a closed profile.




## Adding features

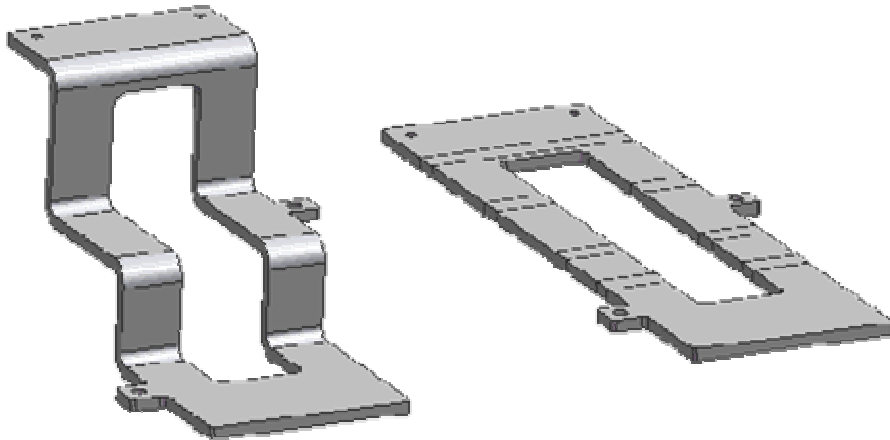
After you have constructed the base feature, you can use the commands on the **NX Sheet Metal** and **Form Features** toolbars to complete the part by adding flanges, jogs, bends, cutouts, holes, pockets, and so on.
















## Creating a flat pattern


You use [Flat Solid](#)  to create a new solid body in the part file while keeping the original (parent).

The flattened body (and flat pattern) are always at the end of at the timestamp order. Every time a new feature is added onto the parent body, the flattened body is placed at the end. It updates to reflect changes in the parent.



-  Model History
- ☒  Fixed Datum Plane (0)
  - ☒  Fixed Datum Axis (1)
  - ☒  Fixed Datum Axis (2)

- ☒  SB Tab (3)
- ☒  Extrude (4)
- ☒  SB Bend (5)
- ☒  SB Bend (6)
- ☒  SB Bend (7)
- ☒  SB Bend (8)
- ☒  Extrude (9)
- ☒  SB Flat Solid (10)
- ☒  Flat Pattern (11)

You then use the [Flat Pattern](#)  command to create a sheet metal flat pattern for export to a machine tool for manufacturing. This step supplements **Flat Solid** by including extra entities such as bend centerlines, tangent lines, and other attributes that provide special machine instructions. You use **Customer Defaults** to specify which entities you want represented, and how you want them annotated.