

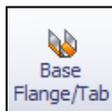
Sheet Metal

Description: Aids in the design of sheet metal part or parts. The sheet metal feature uses open, closed, or multiple closed sketches to create the basic framework or outline of the sheet metal part. This feature also allows the user to modify the part by welding, cutting material, adding material, adding corner treatments, and miters. The sheet metal feature can also build forming tools to help modify future sheet metal parts.

Sheet Metal Toolbar At A Glance

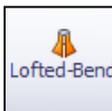


Sheet Metal Tools



Base Flange/Tab:

The initial feature used that designates a new part as a sheet metal part. **Left clicking** on this feature will enable it and prompt the user for a sketch plane or preexisting sketch. Sketches can be open, closed, or contain multiple enclosed sketch. The user can designate the thickness and bend radius in the property manager.



Lofted-Bend:

Employs two **open** sketch profiles and connects them by a loft. The user can specify the thickness of the metal and the bend allowance. *Note: This feature cannot be mirrored.*



Edge Flange:

Combines a bend and sheet metal tab in a single operation. User selects the desired edge, bend radius, angle, flange length, and flange position.



Miter Flange:

Allows the user to add multiple flanges to one or more of the edges of the sheet metal part.



Hem:

Folds back the edge of a sheet metal part to create a smooth edge. The types of hems include: closed, open, tear drop, and rolled.



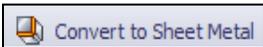
Jog:

Allows the user to add material to a sheet metal part from a sketched line. This feature creates two bends from a single sketched line.



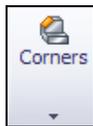
Sketched Bend:

Add a bend to a sheet metal part from a sketch line or lines. The sketched bend property manager allows the user to alter and choose the desired bend position, angle, and gauge radius.



Convert to Sheet Metal:

Create a sheet metal part from a solid body or surface. *Note: If the part is not a constant thickness the sheet metal features may not work properly.*



Corners:

Apply different type of corner treatments to the sheet metal part. Options include: closed contour, welded, and break/trim corner. To chose



Forming Tool:

Create a die that bends, stretches, or forms a sheet metal part. Forming tools can aid in producing louvers, lances, flanges, and ribs. SolidWorks includes some forming tool features



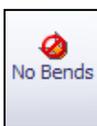
Fold/Unfold:

Flatten or bend one, many, or all of the bends in a sheet metal part.



Flatten:

Temporarily flatten a sheet metal part by activating this feature. Flatten can also be found in the design tree whenever a sheet metal part is being created. At any time the user can unsupress (flatten) or supress this feature leaving it in a three dimensional state. To activate **right click** on the feature or **left click** flatten in the design tree and select **unsuppress**.



No Bends:

Allows the user to roll back all bends in a sheet metal part so that additional features may be added easily. To restore all bends **left click** the feautre again.



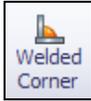
Rip:

Creates a rip in a sheet metal part from either linear sketch entities, model edges, or a combination of both.



Insert Bends:

Flattens the sheet metal part and inserts bends where user desires.



Welded Corner:

Allows the user to add a weld to a sheet metal corner such as a miter, flange, edge flange, and a closed corner.
