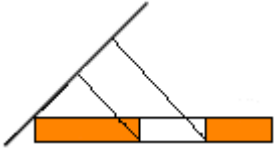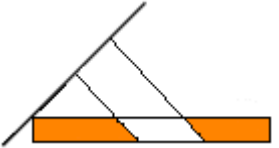# About Sheet Metal Cuts

A sheet metal cut removes material from your part. The cut is made normal to the sheet metal surface, as if the part were completely flat, even if it is in a bent state. The cut adopts the sheet metal material's natural behavior, like bending and warping, when the part is bent.
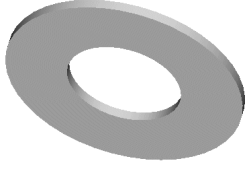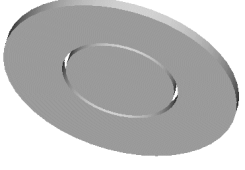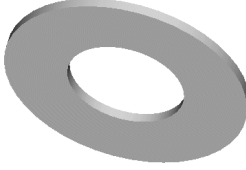
You sketch cuts on a plane and project them onto the sheet metal wall. Either the driving or offset side of the sheet metal wall can drive the cut direction.

Sheet Metal Cut Behavior	Solid Cut Behavior

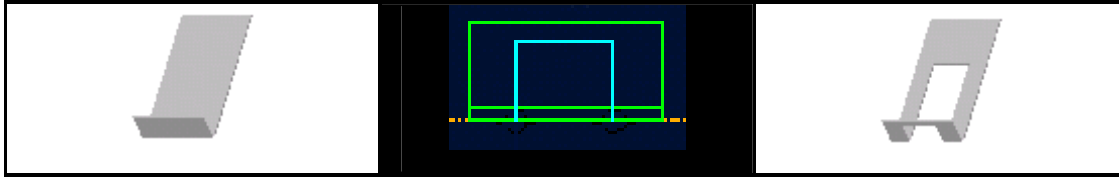You can create three types of cuts:

- **Sheet Metal Cut (solid)**—Removes solid sections of the sheet metal wall.

- **Sheet Metal Cut (thin)**—Removes only a thin section of material, like a thin cut made with a laser.

- **Solid-Class Cut**—Removes solid sections of the sheet metal wall. You can extrude, revolve, sweep, blend, use quilts and make advanced solid-class cuts. To make a defined-angle cut, you must use the solid-class cut. Solid-class cuts can be made on an edge. See the **Part Modeling** Functional Area for information about solid-class cuts.

   Note: Always use the sheet metal-class cut, unless you need tapered edges.

Sheet Metal Cut (Solid)	Sheet Metal Cut (Thin)	Solid-Class Cut

Because sheet metal cuts are surface cuts, you cannot make a cut to partially remove wall thickness. For example, you can not cut a 1cm deep hole in a 10cm thick wall. This may make the Blind depth command somewhat difficult to understand. The Blind depth command applies to cutting on bends. You can sketch the cut to the edge of the bend and project it a blind depth down the bent wall, saving you the time of unbending the wall, making the cut, and bending back the wall.

Original Wall	Blind Cut Sketch	Blind Cut

Note:

- A cut cannot cross two bend lines.

- A cut can never be made on an edge.

- Cutting on angles or bend areas might require a larger dimension scale for proper clearance.

- Cuts can be used to create notch and punch UDFs.

See Also