Sheet Metal

Convert Solid to Sheet Metal ~ 2009
A solid part can be converted directly into a flattened sheet metal version. This is especially useful when a draft solid design is needed for a part that will later be manufactured as sheet metal.

Cross Break ~ 2009
You can now add reinforcement cross breaks to sheet metal parts to stiffen the design, deflect water, and so on. Graphical cross breaks are important design details for downstream manufacturing operations.

Multibody Sheet Metal Parts ~ 2010
SolidWorks multibody part functionality has been extended to sheet metal to allow you to create complex sheet metal designs. Multibody sheet metal parts can consist of multiple sheet metal bodies or a combination of sheet metal and other bodies such as weldment bodies.

Closed Corner Improvement ~ 2010
You can now use the existing Closed Corner command with a greater variety of sheet metal parts.
Flat Pattern Improvements ~ 2010
Graphical highlighting of self-intersecting flat patterns makes it easier to recognize areas that need improvement for manufacturing.

You can also turn off the Normal cut option. This minimizes the weldment gap for manufacturing of rolled sheet metal parts with cut extrudes and produces a flat pattern with different outlines for the inner and outer faces of cut.

Exporting Sheet Metal Parts to DXF or DWG ~ 2010
The new DXF/DWG PropertyManager exports sheet metal bodies to .dxf or .dwg files.
Exporting flat Pattern ~ 2010
The flat pattern of a sheet metal part can be exported as DXF file without having create a drawing first.

Edge Flanges on Curved Planar edge ~ 2010
You can add curved edge flanges to edges flanges created off a Planer base.

Lofted Bend with Bend line ~ 2011
Bend lines are displayed with lofted sheet metal pattern.
Cuts Across Bends ~ 2012
You can unfold a part that has cuts across bends when the cuts do not go through the entire part. This is helpful when making flat patterns for manufacturing.

Swept Flanges ~ 2012
You can create compound bends in sheet metal parts using the Swept Flange tool.

The Swept Flange tool is similar to the Sweep tool; you need a profile and path to create the flange. To create a swept flange, you need an open sketch as the profile, and an open profile path or a series of existing edges in a sheet metal part.

Sheet Metal Costing ~ 2012
The SolidWorks Costing tool helps you calculate how much it costs to manufacture sheet metal parts by automating the cost estimation and quotation process.

To use this tool, the part must contain sheet metal features such as flanges, bends, and forming tools. Features such as holes and cuts are recognized as manufacturing cut paths for operations such as laser, water jet, and plasma cutting.