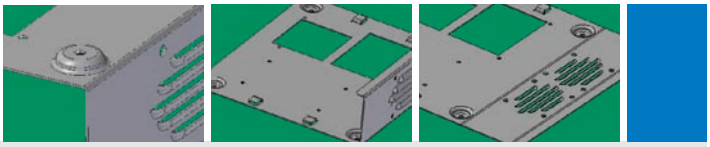
Velocity Series

Intelligent, process-specific sheet metal environment speeds time to manufacturing

www.siemens.com/solidedge

white paper



A core design capability of Solid Edge® software, the Sheet Metal environment provides an entire design-through-fabrication workflow, using streamlined modeling commands that are tailored to the unique needs of sheet metal design.

PLM Software

Answers for industry.

SIEMENS

Solid Edge Sheet Metal Design

Table of contents

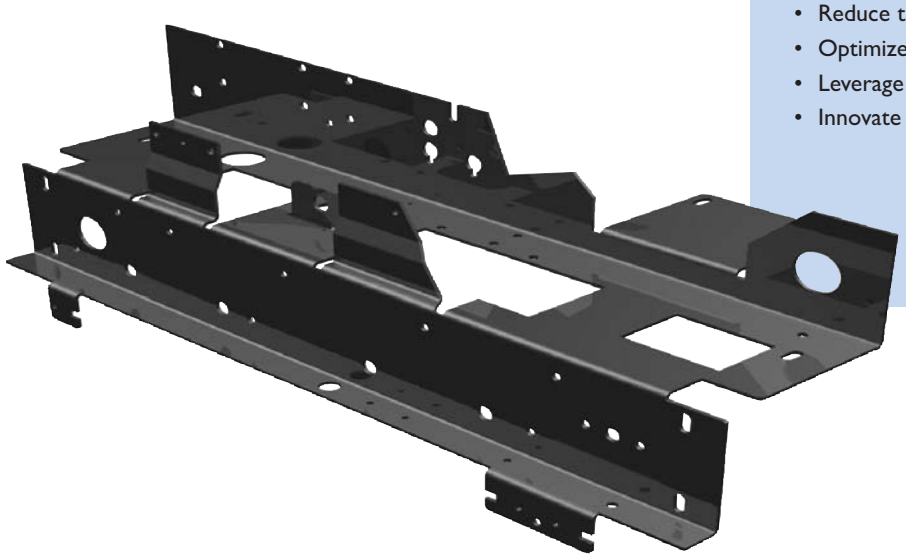
Overview

If you design sheet metal parts, you face some unique challenges. Although parts are typically designed in their "formed" state, they begin as a flat plate so manufacturability becomes a critical aspect of every feature making up the finished part. Add to this the need to account for material thickness, along with bend and corner relief, miters and deformation features and it becomes obvious that you need a highly specialized set of tools if real productivity and quality gains are to be realized.

The advantages of 3D modeling software are well documented. However, CAD programs often take a generic approach to modeling sheet metal parts, forcing you to use workflows, commands and features more appropriate for machined, cast or molded parts.

This paper discusses how a careful consideration of these limitations, combined with extensive research into the processes of manufacturers whose products include sheet metal components, led to the development of the most advanced sheet metal modeling capabilities available – Solid Edge Sheet Metal.

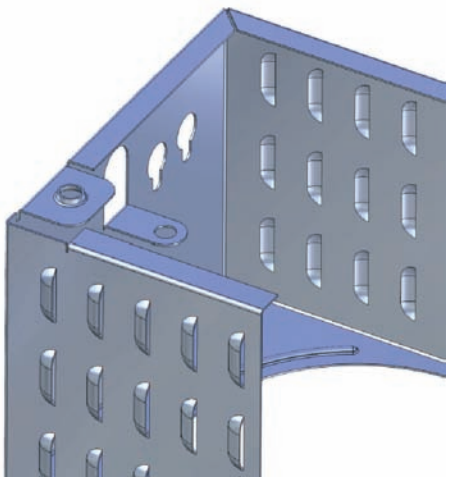
A core design capability of Solid Edge, the Sheet Metal environment provides an entire design-through-fabrication workflow, using streamlined modeling commands that are tailored to the unique needs of sheet metal design.



Road to innovation

If you want your company to be a market leader, you need to innovate relentlessly. Today's most forward-thinking companies are continuously optimizing and improving products and processes – and organizing their value chain so that innovation can flourish. By allowing distributed value chains to work together as a single enterprise, these "innovation networks" drive today's crucial business imperatives. To achieve this level of collaboration companies must find new sources of innovation that extend well beyond in-house invention. Therefore more and more best-in-class companies are choosing new roads to innovation. Solid Edge is the most cost effective, complete and open solution for 2D and 3D design and collaboration, helping you meet these key business requirements:

- Accelerate top-line growth
- Contain costs
- Reduce time-to-market
- Optimize resources
- Leverage globalization
- Innovate more



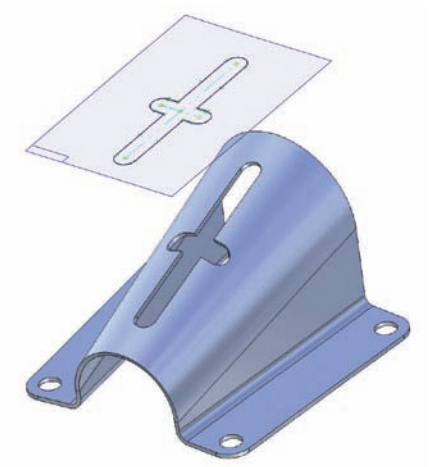
Built-in support for the sheet metal process

Solid Edge's specialized process-specific commands streamline the design of sheet metal components by using terminology you are familiar with and requiring significantly fewer steps.

You enter part properties – material, thickness, bend relief and bend equations etc. – in one place. Your sheet metal parts automatically adhere to these predefined material specifications without having to define new properties each time you add a new feature.

Easy click-and-drag operations quickly add tabs and flanges to the base model, automatically accounting for material thickness to maintain critical internal and external dimensions. Push button options control flange width and location along the part edge.

You can control whether bend relief or corner relief is included as part of the feature and have full control over size and shape of the relief. If you need to change the material thickness, a simple edit to one value will change the entire part. You can also change properties for an individual feature, perhaps identifying one flange as needing a larger bend radius.

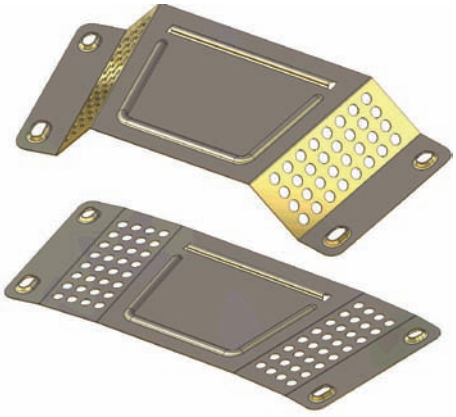


Intelligent sheet metal features

Solid Edge Sheet Metal understands the unique challenges of working with sheet metal parts. When cutouts or holes lie across a bend, a traditional cutout command would result in non-perpendicular faces. By contrast, Solid Edge's normal cutout command creates accurate perpendicular faces, reflecting the fact that the feature would likely be manufactured while flat, then folded.

You can easily design transitional sheet metal shapes and Solid Edge Sheet Metal lets you quickly model features that are manufactured with metal deformation techniques, like deep drawing and coining. Deformation features, such as louvers and beads, are constructed using a single, linear element and simply defining height and depth and whether you want the louver ends formed or lanced.

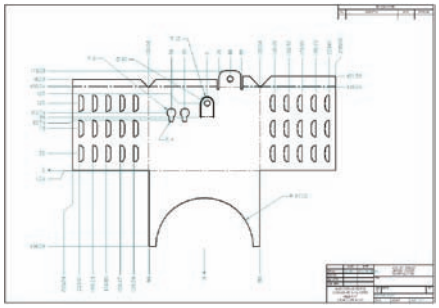
With these associative feature parameters, you can easily make style changes, such as changing a lanced-end louver to a formed-end louver. A more generic CAD approach – such as using library features – would often force you to start over.



Ensuring manufacturability

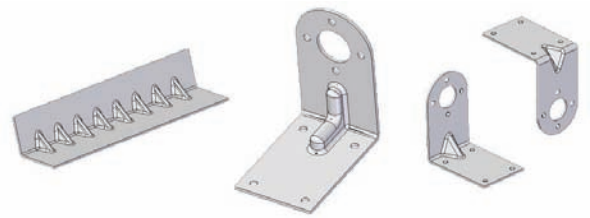
Model validation is another area where Solid Edge is unmatched. A classic manufacturing issue involves cutouts or flanges placed too close in proximity to other cutouts or bends. Typically there is a minimum distance between bends that must be maintained to accommodate the bend die. Designers are aware of this but often don't measure each bend for validation. Solid Edge includes design sensors that handle this operation automatically.

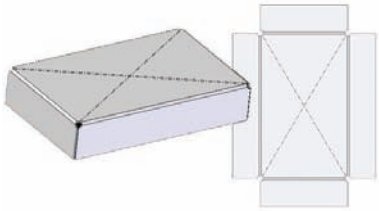
And Solid Edge flattens and rebends sheet metal models, automatically calculating bends from standard or custom formulas. In addition to effortlessly creating an accurate pattern for manufacturing, this simplifies the modeling of cutouts and holes that lie across bends.

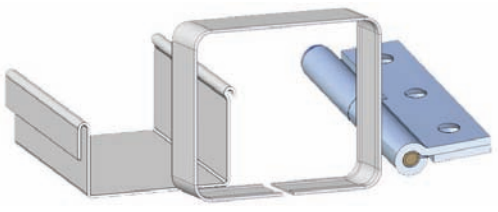


Highly productive drafting

Solid Edge streamlines drawing creation with the industry's most productive drafting system. Formed and flattened components can be detailed and dimensioned and remain associative so they automatically update when you change your design. Innovative tools for shaded views, exploded assemblies, detail and section views, hole tables and coordinate dimensioning all ensure that you represent your parts in the best possible way for customers and manufacturing.



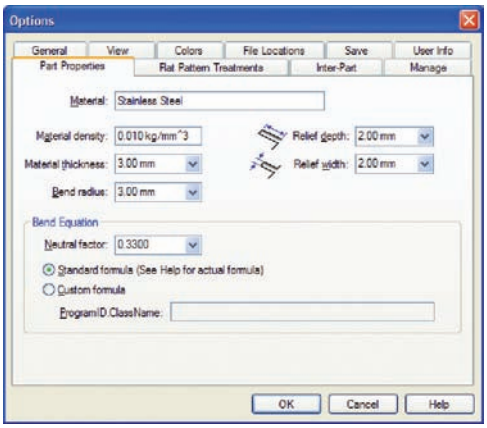




Unmatched sheet metal design productivity

Solid Edge's specialized sheet metal design aids deliver significant productivity gains compared to general-purpose CAD tools. Process-specific commands and structured workflows speed the modeling of sheet metal parts. Built-in intelligence saves additional time by automatically calculating material treatments and validating parts for manufacturability, while manufacturing-ready flat patterns help to eliminate scrap and rework. The result is faster time to manufacturing, backed by improved quality of the sheet metal components.

And Solid Edge Sheet Metal can become a critical component to winning business. It benefits the entire organization by making design information easily accessible to people in marketing, sales, technical publications and so on. Solid Edge Sheet Metal is a comprehensive solution that addresses the key requirements for success in the sheet metal industry today: the ability to ease the design of increasingly complex products by creating sheet metal parts quickly and accurately.

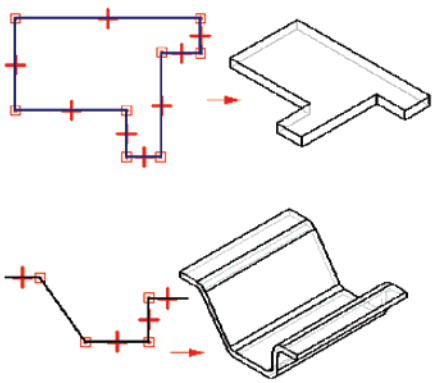


The remainder of this document explores some of the specific commands and workflows that make Solid Edge such a powerful tool for sheet metal design.

Defining part properties

The part properties tab on the options dialog box allows you to set up the sheet metal properties for the part you are constructing. These properties are copied to the variable table and are used each time you create a new feature. For example, when you construct a flange, the material thickness and bend radius are automatically applied.

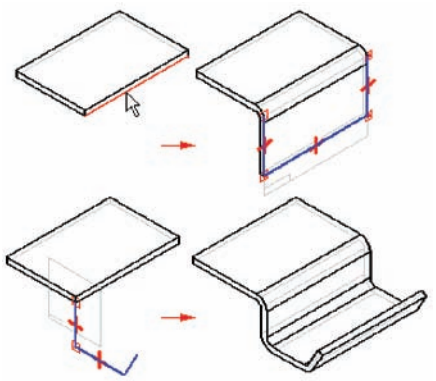
If you edit these values later, the part will update. For example, if the material thickness for the part needs to be increased, you can edit the value for the material thickness and the material thickness for the entire part will change.



Starting with a base feature

Similar to working in the Solid Edge Part environment, you construct a sheet metal part by constructing a base feature and then adding features until the part is complete. Sheet metal properties such as material thickness, bend radius and bend relief are easy to control and manage. Instead of defining these properties each time you add a new feature, default property values are provided in the template.

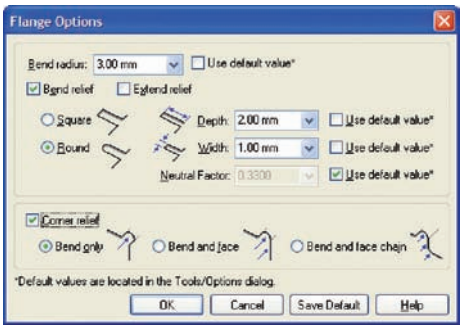
You can construct a base feature with the tab, contour flange or lofted flange commands. The tab command allows you to construct a flat feature of any shape using a closed profile. The contour flange command allows you to construct a feature comprised of one or more bends using an open profile. The lofted flange command allows you to quickly construct a transitional shape using two open profiles on parallel reference planes. Like the contour flange command, the lofted flange command automatically adds bends using the bend radius property. You do not have to draw an arc at each bend location.



Adding features

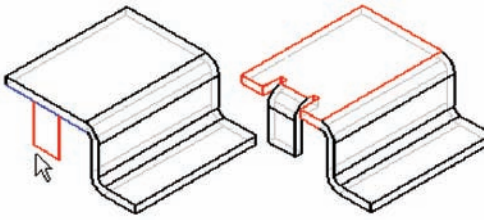
After you have constructed the base feature, you can use the commands on the Sheet Metal features toolbar to complete the part by adding features such as flanges, cutouts, holes, louvers, beads and so forth.

Other programs rely on a library-based system to develop these features, but they limit users to editing only the dimensions of the existing shapes. Solid Edge also has many time saving features to speed up design in the context of an assembly. For example, flanges can be associatively created or modified to match the face of an existing part in the assembly, and will automatically update if that face changes.



Feature properties

You can also change properties for an individual feature. For example, you may want to use one bend radius value for the entire part, except for one flange, which needs a larger bend radius. By simply selecting the flange you wish to edit and clicking flange options on the ribbon bar, you can change the bend radius property.

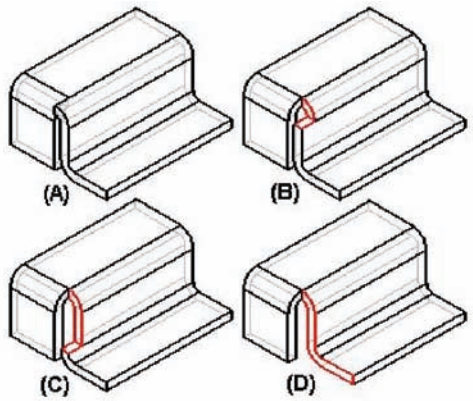


Bend relief and corner relief

When constructing and modifying flanges and contour flanges, you can use the flange options dialog box to control whether bend relief or corner relief is included as part of the feature. If you define a bend or corner relief, you can also control the size and shape.

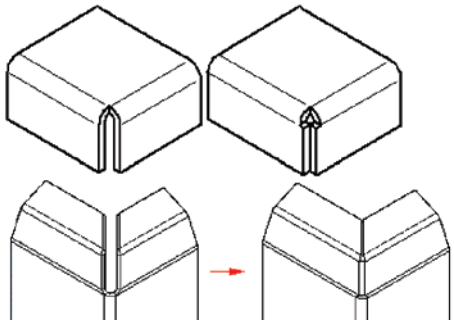
When specifying bend relief, it is applied to the source face from which the flange is constructed. For example, when constructing a partial flange centered on an edge, bend relief is added to the source face on both sides of the flange.

You can also use the extend relief option to specify whether the bend relief is applied only to the area adjacent to the bend or to the entire source face.



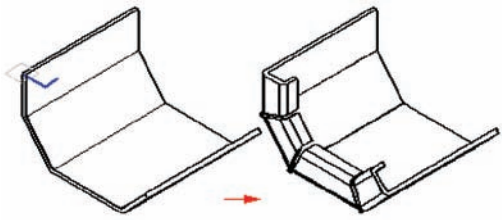
You can define the following options when applying corner relief:

(A) None
(B) Bend only
(C) Bend and face
(D) Bend and face chain



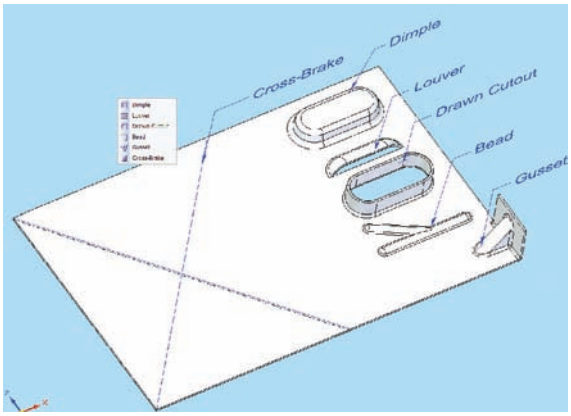
Closing corners

The close corner command will modify two flanges in one operation to close the corner where the two flanges meet. Since closed corner is a treatment, you do not have to draw a profile; just select the edges you want to modify. The close corner command allows for overlaps, gaps, three-bend and watertight corners – a range of options unmatched by any other program.



Mitering contour flanges

You can miter the ends of a contour flange by setting options on the miter tab of the contour flange options dialog box. For example, when constructing two contour flanges that will overlap, you can miter the ends where the flanges meet.



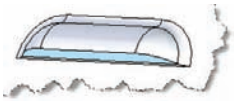
Deformation features

Standard sheet metal features usually either add or remove materials; however such is the nature of sheet metal fabrication, that it is sometimes easier, less work and faster to simply deform the existing sheet metal surface to either remove sharp edges, stiffen a surface or to add strength to a face or corner.

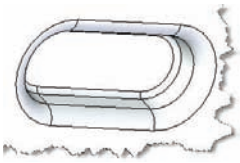
Solid Edge uses process-specific deformation features to dramatically speed up your design process and free engineers from mundane detailing tasks. Complex modeling scenarios can be achieved using deformation features, just as easily as adding a hole.

Deformation features (louvers, dimples, gussets, hems and beads) not only speed up sheet metal design time, by automatically assigning essential design parameters, such as bend, punch and die radii, and corner treatments in a single feature but also ensure each design is consistent throughout the whole process making sure your design is manufacturable and meets other key design criteria. For example, with the louver command, you describe the location with a single straight line and enter the height and width for this feature, then Solid Edge takes care of punch and die radius, material thickness and calculates all end conditions automatically. Changing these parameters is quick and easy at any time for individual features, multiple features or the complete model.

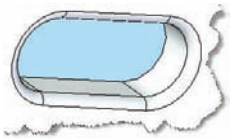
Deformation features used by Solid Edge are:



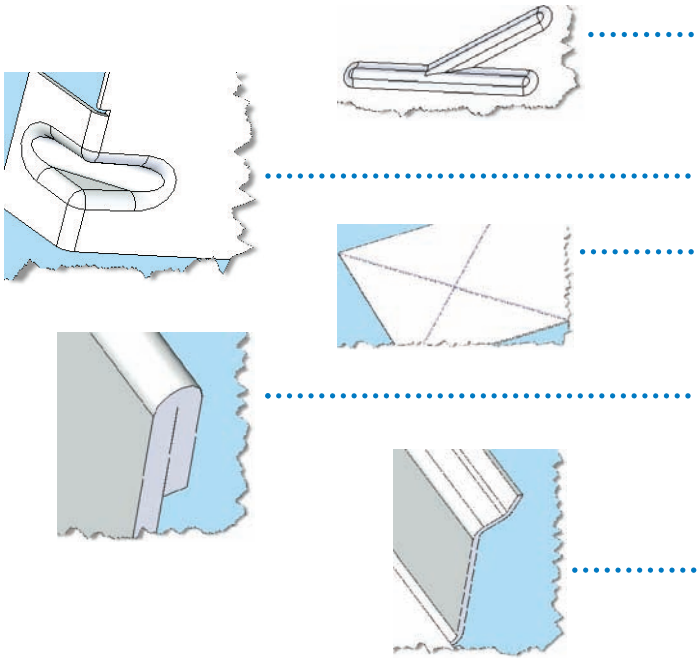
Louvers – are a common sheet metal design detail for adding ventilation holes into a panel that are safe, easy to manufacture and look tidy. After positioning a louver and setting its size, Solid Edge automatically adds die and punch radii, and allows options for open (lanced) ends or closed-end conditions.



Dimples – are commonly used to raise a panel face, add stiffness to sheet metal designs. After drawing a profile, Solid Edge has options to automatically add taper angle, rounding sharp corners and punch and die radius.



Drawn cutouts – are similar to dimples but the top face is removed. Drawn cutouts are often used in thin sheet metal where fasteners are required. The excess material or burr is usually sufficient to add a thread and remove the need to add a weld nut. This feature is also common for adding glazed or plexiglas inspection panels etc.



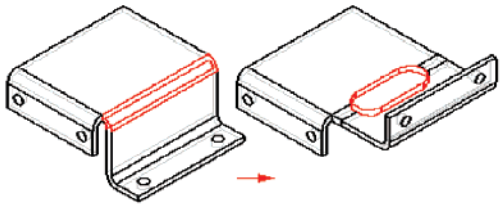
Beads – or swages are often used to add stiffness to large flat areas of sheet metal panels. After simply drawing a centerline for the profile, Solid Edge automatically adds the bead options, such as bead profile, punch and die radii and open- or closed-end conditions.

Gussets – add a reverse pressing across a bend to add substantial strength and stiffen sheet metal corners (corner stiffener).

Cross-Brakes – are often used to stiffen large unsupported sheet metal surfaces; they are often seen in sheet metal duct work and machine guards. Brake lines are represented in the sheet metal flat, DXF flat patterns and 2D drawing views.

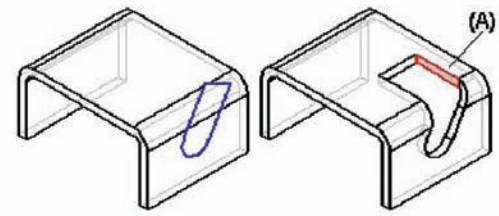
Hems (commonly referred to as a safe edges) – provide more capabilities than simply folding a corner flattening the flange. Options are provided to create rolled and circular profiles; for example a wired edge or hinge can be easily modeled. Hems can be applied to any combination of straight or curved edges.

Contour flanges – can be applied to straight and curved edges, with options for corner treatment, depending on how the part will be formed.



Adding features across bends

On many sheet metal parts, cutouts or holes lie across a bend. You can use the unbend command to flatten a flange so you can add a feature, such as a cutout or a simple hole, across the bend. After you add the feature, you can use the rebend command to rebend the flange.



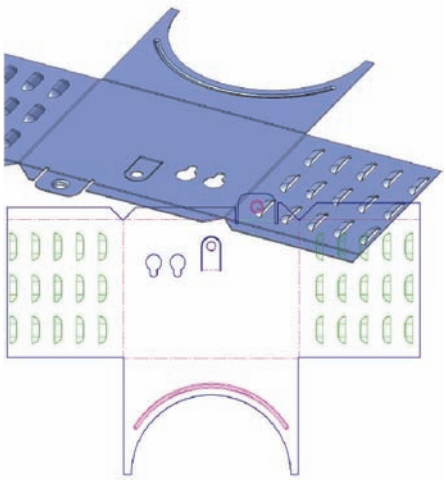
Sheet metal cutouts

When constructing sheet metal parts, you can construct cutouts using the cutout command or a specialized normal cutout command. When you use the normal cutout command to construct the cutout, the software creates thickness faces that are perpendicular (A) to the sheet faces.

This is a good example of a generic modeling feature not being appropriate for sheet metal components. Although a standard cutout command would successfully construct the cutout, you might not be able to flatten the part later. Even if you could successfully flatten the part, the cutout would contain non-perpendicular faces and notches at the transitions between flat and bend. The normal cutout feature overcomes each of these problems and better reflects that the feature would likely be manufactured while flat. For text cutouts, Solid Edge provides punchable font types used for many sheet metal designs. This means the internal shapes for letters such as O, B and D remain intact and manufacturable.

Adding features in the part environment

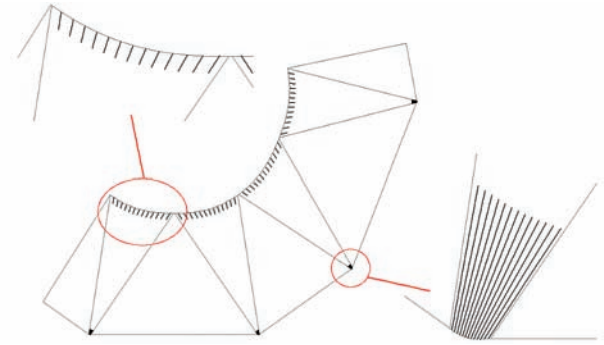
You can access a Sheet Metal document (.PSM) in the part environment, in order to add non-sheet metal features, using switch to part environment. After you have finished, you can return to the Sheet Metal environment using switch to Sheet Metal environment.



Manufacturing support

Manufacturing support is an area where Solid Edge Sheet Metal excels. Its save flat as DXF and flat pattern generation commands allow the user to create a CAM-ready flat pattern DXF file directly from the sheet metal model without the need to create a drawing first.

There is a thin line between sheet metal design and manufacturing responsibilities, more so than most design disciplines. Where the designer leaves off and the production engineer takes over is often dictated by the size of the company. Many times it is the design engineer who has to decide how components need to be made. We at Siemens PLM Software recognize this scenario and provide manufacturing support that can be completed at the design or manufacturing stages.



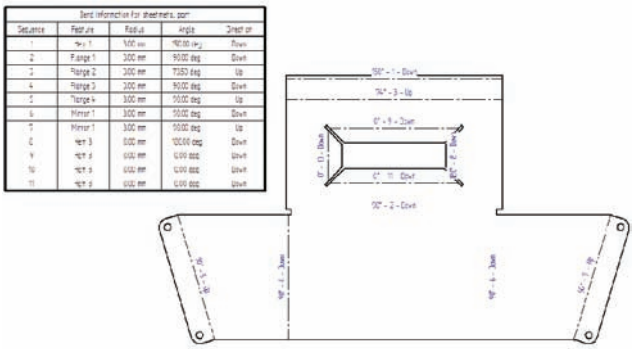
Bend tables in part – allow a way of documenting the bend order for a component while allowing the designer freedom to design the part the best way.

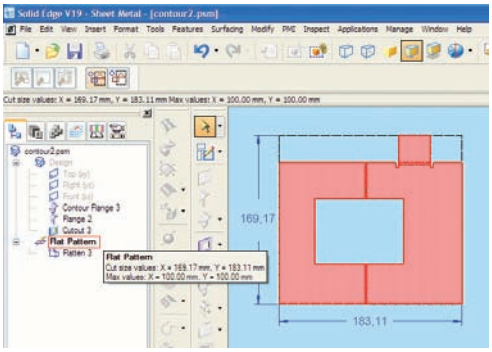
Flat pattern cut size – reports the maximum blank size for unfolded components. They can be linked to a new sensor that will warn if the raw material sheet size is too small.

Option to 'unbend' all bends – means designers no longer have to select all bends first if they want to unfold the whole part.

Write bend data to DXF files – provides control over elements to represent up and down bends, bend lines and bend angles etc.)

Triangulation lines for conical bends – an important enhancement for anyone who creates transitional sheet metal work like square to rounds or conical shapes. These shapes are often formed using a press brake. Designers can decide how to bisect the developed pattern, and center lines will be added to the flat pattern.

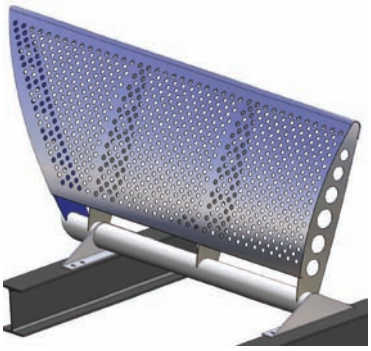




With these commands, Solid Edge has automated perhaps the most widely used workflow – that of creating manufacturing-ready files. When a flat pattern is generated, Solid Edge automatically combines colinear lines into a single line. Options are also provided to convert spline-based curves produced by corner relief into lines, using a supplied tolerance. The settings are used for all flat pattern generation methods and are intended to provide an NC-ready flat pattern for downstream manufacturing.

Users can automatically add corner treatment to prevent dwell burning caused by laser manufacturing machines.

A number of options are available for generating flat patterns and Solid Edge allows layer mapping, so manufacturing software operators can open a Solid Edge file directly and begin showing critical bend lines or other useful information as each element type is carefully mapped to an appropriate layer.



Conclusion

Solid Edge streamlines the entire sheet metal product development process, from design of sheet metal components, through flat pattern development and the creation of engineering drawings. Along with integrated partner applications for analysis, nesting, NC programming and related tasks, Solid Edge is the foundation of a complete design-through-fabrication solution that helps shrink product lead times, improve quality and reduce costs.

About Siemens PLM Software

Siemens PLM Software, a business unit of the Siemens Industry Automation Division, is a leading global provider of product lifecycle management (PLM) software and services with 5.5 million licensed seats and 51,000 customers worldwide. Headquartered in Plano, Texas, Siemens PLM Software's open enterprise solutions enable a world where organizations and their partners collaborate through Global Innovation Networks to deliver world-class products and services. For more information on Siemens PLM Software products and services, visit www.siemens.com/plm.

For more information, contact your local Solid Edge representative:

Destek Yazılım Ltd. Şti.
Barbaros Mah. Denizmen Sk.
No:4/2 Koşuyolu – Usküdar
34662 Istanbul
Tel: (0216) 651 98 01
Fax: (0216) 651 98 02
www.destek-yazilim.com

SIEMENS

Division headquarters

United States
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
972 987 3000
Fax 972 987 3398

Solid Edge – www.siemens.com/solidedge

Americas
800 807 2200
Fax 314 264 8922

Europe
44 (0) 1202 243455
Fax 44 (0) 1202 243465

Asia-Pacific
852 2230 3308
Fax 852 2230 3210

5/08