Sheet Metal Overview

This chapter describes the terminology, design methods, and fundamental tools used in the design of sheet metal parts. Building upon these foundational elements of design, you can quickly and efficiently design your sheet metal parts with Autodesk® Inventor®.

Chapter Objectives

After completing this chapter, you will be able to:

- Describe common sheet metal concepts, terms, and manufacturing equipment, and use Autodesk Inventor to design a simple sheet metal part.
- Describe the two methods for designing sheet metal parts and use them to create sheet metal designs in Autodesk Inventor.
- Create sheet metal rules incorporating style and materials information for common sheet metal parts.
Introduction to Sheet Metal

This lesson describes sheet metal concepts, terminology, and manufacturing equipment, and provides an overview of designing sheet metal parts with Autodesk Inventor.

Sheet metal design differs from traditional mechanical design in several ways. For example, when you design a sheet metal component, you must create a flat pattern and design the part in such a way that it can be folded or bent. You must also consider how the part will be manufactured.

In the following illustration, a typical sheet metal part is shown in the 3D folded state and the flat pattern layout. Bend lines (1) are displayed in the flat pattern.

Objectives

After completing this lesson, you will be able to:

- Explain basic sheet metal concepts and common terms used in the sheet metal industry.
- Describe various types of manufacturing equipment that is common in a sheet metal manufacturing environment.
- Describe the overall process for designing sheet metal parts in Autodesk Inventor.
Sheet Metal Concepts and Terminology

Sheet metal design requires the implementation of specific methods and concepts. For example, each sheet metal part requires the generation of a flat pattern to represent the part in its unfolded state. To generate this flat pattern, you must consider the type and thickness of the material, grain direction, and machine setup for bending in order to develop standard bend allowance or K-factor values that are used in calculating the flat pattern.

Most sheet metal shops have standards in place that the designer can use while generating the flat pattern.

The following illustration shows the Sheet Metal Defaults dialog box. Use Sheet Metal Defaults to set the current Sheet Metal Rule, Material Style, and Unfolding Rule.

Sheet Metal Concepts Defined

The following list contains some requirements that are common to all sheet metal designs.

- Definition of a sheet metal rule.
- Sketch geometry.
- Using standard sheet metal tools.
- Creating and editing the flat pattern layout.
- Creating 2D documentation that represents the 3D folded model and the flat pattern layout.
### Common Sheet Metal Terms

<table>
<thead>
<tr>
<th>Term</th>
<th>Definition</th>
<th>Illustration</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Neutral axis</strong></td>
<td>Theoretical axis passing through the sheet metal part. Generally assumed to be .44 of the thickness from the inside surface of the metal. The metal along this axis does not stretch or compress during bending.</td>
<td><img src="image" alt="Neutral Axis" /></td>
</tr>
<tr>
<td><strong>K-factor</strong></td>
<td>Represents the theoretical position of the neutral axis.</td>
<td><img src="image" alt="K-Factor" /></td>
</tr>
<tr>
<td><strong>Bend allowance</strong></td>
<td>The distance around the bend measured along the neutral axis. The developed flat length is calculated by using the formula $L1 + L2 + BA = \text{flat length}$.</td>
<td><img src="image" alt="Bend Allowance" /></td>
</tr>
<tr>
<td>Term</td>
<td>Definition</td>
<td>Illustration</td>
</tr>
<tr>
<td>--------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>--------------</td>
</tr>
<tr>
<td>Bend radius</td>
<td>Inside radius of the bend.</td>
<td><img src="image1" alt="Bend Radius" /></td>
</tr>
<tr>
<td>Bend relief</td>
<td>Area next to a bend where material is removed to prevent stressing or tearing while bending.</td>
<td><img src="image2" alt="Bend Relief" /></td>
</tr>
<tr>
<td>Flat pattern</td>
<td>Representation of the sheet metal part before it is bent. The developed length is calculated according to the current bend allowance or K-factor setting.</td>
<td><img src="image3" alt="Flat Pattern" /></td>
</tr>
</tbody>
</table>
Bend Allowances Defined

The following list describes items to consider when working with bend allowances.

- Bend allowances can vary from shop to shop and machine to machine.
- Bend allowances can vary with the type of material.
- Bend allowances can vary depending on whether the bend is going with or against the grain of the material.
- For high-precision sheet metal work, the bend allowance can vary with each sheet of steel.
- If you create designs for different manufacturing shops, you can create multiple bend tables or sheet metal styles to reflect the different bend allowances.

Example of a Folded Model and its Flat Pattern

Until recently, flat pattern layouts were created from the folded model which would require you to perform elaborate calculations manually or by using special formulas in a computer program. Today, your CAD system automatically generates the flat pattern based on user input.

In the following illustration, you see a folded model and the automatically generated flat pattern side by side.
Sheet Metal Manufacturing Equipment

This section describes the technologies and equipment used in sheet metal manufacturing. In order to effectively design sheet metal parts, you must be familiar with the various equipment that is used to produce the parts and also with their related technologies.

In the following illustration, sequential motion is captured to show the formation of a part in a press brake.

Sheet Metal Fabrication Equipment

The following table summarizes common equipment used in the manufacture of sheet metal components.

<table>
<thead>
<tr>
<th>Equipment</th>
<th>Definition</th>
<th>Illustration</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear</td>
<td>Cuts large sheet metal stock into smaller, more manageable sizes for further manufacturing processes.</td>
<td><img src="image" alt="Shear Illustration" /></td>
</tr>
<tr>
<td>Equipment</td>
<td>Definition</td>
<td>Illustration</td>
</tr>
<tr>
<td>-----------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>--------------</td>
</tr>
<tr>
<td><strong>Press brake</strong></td>
<td>Used to bend the flat sheet metal into its final shape.</td>
<td></td>
</tr>
<tr>
<td><strong>Punch press</strong></td>
<td>Punches a predefined shape out of the sheet metal. Can be used to remove material or to form material such as louvers.</td>
<td></td>
</tr>
<tr>
<td><strong>Laser cutter</strong></td>
<td>Cuts sheet metal using a laser beam. The material is melted along the beam path.</td>
<td></td>
</tr>
</tbody>
</table>
### Designing Sheet Metal Parts with Inventor

When you use Autodesk Inventor to design sheet metal parts, you begin with a sketch that represents either the part's initial face or a part profile as it changes direction with each bend. You use standard sheet metal design tools to create 3D features such as a face, flange, cut, and contour flange. You also assign to each part a sheet metal rule which specifies material, material thickness, unfolding rule, bend relief and corner relief properties, and flat pattern punch representation.

As you create the design, you use the Flat Pattern tool to generate the part’s flat pattern. You should generate the flat pattern early in the design process, because it warns you of any potential design problems related to the generation of a valid flat pattern. You complete the design by generating 2D documentation for the 3D part and flat pattern layout.
Sheet Metal Terminology

In the following table, common sheet metal properties, characteristics, parts, and conditions are related to the Autodesk Inventor tools that you use to create and manipulate them.

<table>
<thead>
<tr>
<th>Sheet Metal Terminology</th>
<th>Definition</th>
<th>Autodesk Inventor Tool</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material type/thickness</td>
<td>Specific material properties, unfold options, and flat pattern punch representation.</td>
<td>Sheet Metal Defaults</td>
</tr>
<tr>
<td>Bend radius</td>
<td>Inside radius of the bend.</td>
<td>Dialog Box &gt; Bend tab</td>
</tr>
<tr>
<td>Bend relief</td>
<td>Used to prevent stressing or tearing when the part is bent.</td>
<td>Dialog Box &gt; Bend tab</td>
</tr>
<tr>
<td>Sheet Metal Terminology</td>
<td>Definition</td>
<td>Autodesk Inventor Tool</td>
</tr>
<tr>
<td>-------------------------</td>
<td>------------</td>
<td>-----------------------</td>
</tr>
<tr>
<td>Flat pattern</td>
<td>Representation of the sheet metal part before bending.</td>
<td>Flat Pattern</td>
</tr>
<tr>
<td>Face</td>
<td>A flat area of the sheet metal part that is used to generate the orientation of the flat pattern.</td>
<td>Face</td>
</tr>
<tr>
<td>Bend</td>
<td>The radius area between the flat surfaces of the sheet metal part.</td>
<td>Bend</td>
</tr>
<tr>
<td>Flange</td>
<td>Material added to the edge of a sheet metal part.</td>
<td>Flange</td>
</tr>
<tr>
<td>Contour</td>
<td>The profile of a sheet metal part, including all bends and faces.</td>
<td>Contour Flange</td>
</tr>
<tr>
<td>Hem</td>
<td>Added to the edge of a sheet metal part for strength or to hide sharp edges.</td>
<td>Hem</td>
</tr>
<tr>
<td>Corner</td>
<td>A condition that determines how adjoining flanges meet.</td>
<td>Corner Seam</td>
</tr>
<tr>
<td>Punch</td>
<td>Removes material of a predefined shape from the sheet metal part.</td>
<td>Punch Tool</td>
</tr>
</tbody>
</table>
Designing Sheet Metal Parts

Without using software applications, sheet metal design might consist only of rough sketches and some dimensions on paper. This method leaves the hard work for the fabrication shop. Many prototypes must be built before the final shape and sizes are determined. By taking full advantage of the tools in the Inventor software, you can design efficiently and accurately, thereby eliminating the need for multiple prototype parts. You can then effectively communicate your design and manufacturing processes for fabrication.

Process: Creating a Sheet Metal Part

The following steps provide an overview of designing sheet metal parts with Autodesk Inventor.

1. Begin a new part using a sheet metal-specific template file. For example, you can choose *Sheet Metal.ipt* in the New File dialog box.

2. Use sketch tools to define the initial profile and then exit the sketch.
3. Use the Sheet Metal Defaults tool to specify your sheet metal rule, material style, and unfolding rule.

4. Use the Face tool to turn your sketch into a sheet metal part, which applies the settings from the active sheet metal rule.

5. Add flanges, punches, and other sheet metal features as needed.

6. Create a flat pattern of your folded model.
7. Create 2D documentation for manufacturing including needed views of the folded model and the flat pattern.
Exercise: Create a Simple Sheet Metal Part

In this exercise, you create a simple sheet metal part and produce a flat pattern view in a drawing.

The completed exercise

Completing the Exercise
To complete the exercise, follow the steps in this book or in the onscreen exercise. In the onscreen list of chapters and exercises, click Chapter 1: Sheet Metal Overview. Click Exercise: Create a Simple Sheet Metal Part.


2. Sketch and constrain a rectangle as shown. Click Finish Sketch to exit the sketch.
3. Click Sheet Metal tab > Setup panel > Sheet Metal Defaults.
   - Clear the Use Thickness from Rule checkbox.
   - For the thickness, enter **2.0 mm**.
   - Click OK.

4. Click Sheet Metal tab > Create panel > Face.
   In the Face dialog box, click OK to accept the defaults.

5. Switch to the Home view. Notice that your part now has thickness.

6. To create a flange on one end:
   - Click Sheet Metal tab > Create panel > Flange.
   - Select the top left part edge as shown.

7. In the Flange dialog box, for Height Extents, enter **85 mm**. Click OK to accept the remaining defaults and create the flange.

8. Click Sheet Metal tab > Modify panel > Corner Round.
   - Select the four corners of the part as shown.
   - In the Corner Round dialog box, enter **3 mm** for the Radius value.
   - Click OK.
9. Click Sheet Metal tab > Flat Pattern panel > Create Flat Pattern. Your flat pattern is created and you are placed in the flat pattern working environment.

10. In the browser, double-click the Folded Model to return to the sheet metal environment. The flat pattern element in the browser is grayed out.

11. To save the part file:
   - Click File menu > Save.
   - In the Save As dialog box, for File Name, enter Door_Bracket.
   - Click Save.


13. To place a flat pattern view in your drawing:
   - Click Place Views tab > Create panel > Base.
   - In the Drawing View dialog box, verify Door_Bracket.ipt is the selected file.
   - Under Sheet Metal View, click Flat Pattern.
   - Click to locate the flat pattern view in the upper left corner of your drawing sheet.

14. Close all files. Do not save.
Lesson: Sheet Metal Design Methods

In this lesson, you learn to use the two common sheet metal part design methods, folded, which is how the part appears after manufacturing, and flat, where the component’s flat pattern is first developed and then folded using Autodesk Inventor.

Designers often have multiple options and methods that they can use to complete their designs. Thorough knowledge of these methods assures that the parts and assemblies are designed well and efficiently.

The following illustration compares the two methods of design. On the top, the part is designed in its folded state, and below, it is designed in the flat.

Objectives
After completing this lesson, you will be able to:

- Explain the two major sheet metal design methods.
- Describe how to design sheet metal parts in the folded or flat pattern states.
Sheet Metal Design Methods

There are two sheet metal design methods that you can use. You can design your part in its folded state or in its flat state. Each design has its benefits, but the folded state is the easier approach.

In the following illustration, on the left, a sheet metal part is modeled in the folded state. On the right, a sheet metal part is modeled in the flat state.

**Design in Folded State Defined**

When you design sheet metal parts in 3D, you use sheet metal tools such as the Face, Flange, Cut, and Hole tools. When you work in the context of an assembly, you can design the sheet metal part using geometry from other parts in the assembly as a reference for the location of features. Designing complex sheet metal parts requires that you use this design method.

The following table summarizes the steps that you follow when designing in the folded state.

<table>
<thead>
<tr>
<th>Tool or Action</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create a sketch of the base face</td>
<td><img src="image" alt="Sketch" /></td>
</tr>
<tr>
<td>Tool or Action</td>
<td>Description</td>
</tr>
<tr>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>Convert the sketch to a sheet metal face</td>
<td></td>
</tr>
<tr>
<td>Create flanges, holes, and other sheet metal features</td>
<td></td>
</tr>
<tr>
<td>Automatically generate flat pattern</td>
<td></td>
</tr>
</tbody>
</table>
Design in Flat State Defined

When you design sheet metal parts in the flat state, you create a 2D representation of the flat pattern and use the Fold tool to bend the sheet metal part. Settings for bend allowance and bend radius are applied automatically to the part as it is folded. The flat pattern remains as dimensioned, and the sizes of the features on the folded version of the part adjust to reflect bend allowances.

The following table summarizes the steps that you follow when designing in the flat state.

<table>
<thead>
<tr>
<th>Tool or Action</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sketch the flat pattern</td>
<td><img src="image" alt="Diagram" /></td>
</tr>
<tr>
<td>Make into a face</td>
<td><img src="image" alt="Diagram" /></td>
</tr>
</tbody>
</table>
## Benefits of Each Design Method

Use the following table as a guide for choosing the most appropriate design method.

<table>
<thead>
<tr>
<th>Design Method</th>
<th>Design in 3D</th>
<th>Design from Flat</th>
</tr>
</thead>
<tbody>
<tr>
<td>Assembly-centric design</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Flat pattern sizes take precedence over final feature sizes</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>Final feature sizes take precedence over flat pattern sizes</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Simple sheet metal parts</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Complex sheet metal parts</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Creating 3D parts from 2D flat patterns (DXF/DWG)</td>
<td>No</td>
<td>Yes</td>
</tr>
</tbody>
</table>
Using Two Different Design Methods

The two design methods used in this lesson are the most common ways to design your sheet metal models. You can also use a combination of both methods.

In the following illustration, row 1 represents designing in the finished state. No additional objects are needed to create a flange.

Row 2 represents designing in the flat state. Sketch geometry was required for the Bend tool to create the lower flange.

Design of Sheet Metal Parts Using Two Methods

Sheet metal designs must communicate accurate information, so that a final product can be created successfully. You can use either of the methods that are presented in this lesson or a combination of the two. If there is a condition that you have a hard time producing in the folded model, you can use tools to produce that specific condition on the automatically generated flat pattern. Modifications to the flat pattern which are performed in this manner are not reflected in the folded model state.
Designing a Sheet Metal Part in Its Finished State

The following steps outline the process of designing a sheet metal part in its finished state.

1. Use the Sheet Metal Defaults tool to specify the sheet metal rule, material style, and unfolding rule according to your design intent.

2. Sketch the first face of the sheet metal part or the contour of the sheet metal profile.

3. Use the Face tool to create the first face of the sheet metal part, or use the Contour Flange tool to create the sheet metal part based on the profile.
4. Use the Autodesk Inventor sheet metal tools to add features such as flanges or additional faces.

5. Use the Autodesk Inventor tools to add features such as holes, cuts, or punches.

6. Use the Flat Pattern tool to generate the flat pattern of the part.
Designing a Sheet Metal Part in the Flat State

The following steps outline the process of designing a sheet metal part from a flat pattern.

1. Use the Sheet Metal Defaults tool to specify the sheet metal rule, material style, and unfolding rule according to your design intent.

![Sheet Metal Defaults dialog box]

2. Sketch the flat pattern.

![Flat pattern sketch]

3. Use the Face tool to create a 3D representation of the flat pattern.
4. For each cut, hole, punch, or other feature, create a new sketch on the face of the part. Use the appropriate tool to create the feature on the 3D representation of the flat pattern.

5. For each bend, create a new sketch and draw a line (1) representing the location of the bend.
   **Note:** The line that represents the bend location must start and end on the edge of the sheet metal part.

6. For each bend on the part, use the Fold tool to create the bend along the selected bend line.
## Design Methods and Tools

The following table lists the various sheet metal tools and the design methods in which they are commonly used.

<table>
<thead>
<tr>
<th>Sheet Metal Tools</th>
<th>Design in 3D (Folded)</th>
<th>Design from Flat</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sheet Metal Defaults</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Flat Pattern</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Face</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Contour Flange</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Cut</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Flange</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Hem</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Fold</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Sheet Metal Tools</td>
<td>Design in 3D (Folded)</td>
<td>Design from Flat</td>
</tr>
<tr>
<td>-------------------</td>
<td>------------------------</td>
<td>------------------</td>
</tr>
<tr>
<td>Corner Seam</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Bend</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Hole</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Corner Round</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Corner Chamfer</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Punch Tool</td>
<td>Yes</td>
<td>Yes</td>
</tr>
</tbody>
</table>
Exercise: Use Sheet Metal Design Methods

In this exercise, you use two different design methods to complete a simple sheet metal part design.

1. Open Z_Bracket.ipt.
2. Activate the Sheet Metal Defaults command.

Completing the Exercise
To complete the exercise, follow the steps in this book or in the onscreen exercise. In the onscreen list of chapters and exercises, click Chapter 1: Sheet Metal Overview. Click Exercise: Use Sheet Metal Design Methods.

3. To modify material thickness and bend radius values:
   - In the Sheet Metal Defaults dialog box, click Edit Sheet Metal Rule.
   - On the Sheet tab, under Sheet, for Thickness (1), enter 1.0 mm.
   - On the Bend tab, under Bend Radius (2), enter Thickness*2.
   - Click Save. Click Done.

4. In the Sheet Metal Defaults dialog box, select Use Thickness From Rule, if it is not selected. Click OK.
5. In the browser, double-click Sketch1.
6. Sketch a rectangle and dimension it as shown.

7. Click Finish Sketch to exit the drawing.
8. Activate the sheet metal Face command. Click OK to accept the defaults and create your sheet metal base face.

9. Click View tab > Appearance panel > Color Override list > Blue Pastel.

10. To create a flange using the Design in Finished State method:
   - Activate the sheet metal Flange command.
   - Click to select the top left edge as shown.

11. In the Flange dialog box:
   - Under Height Extents, click Distance. Enter 15 mm.
   - Under Height Datum, click the first option, Bend from the Intersection of the Two Outer Faces.
   - Under Bend Position, click the second option, Bend from the Adjacent Face.
   - Click OK to create the flange.
12. To create a flange using the Design in Flat State method:
   - Create a new sketch on the base face.
   - Project existing part edges as needed.
   - Sketch and constrain a line as shown.
   - Click Finish Sketch.
   **Note:** Make sure that the line begins and ends on the edge of the part.

13. To bend the part:
   - Activate the sheet metal Fold command.
   - For the bend line, click your sketch line.
   - Adjust the Flip Controls and Fold Location as needed to match the bend preview as shown.
   **Note:** Clicking the highlighted buttons produces the correct result.

14. Click OK to complete the bend.

15. Close all files. Do not save.
Lesson: Sheet Metal Rules

This lesson describes the purpose and use of sheet metal rules. The ability to control sheet metal properties in the same fashion as you do Inventor styles provides a consistent workflow and enables you to share sheet metal properties company-wide through use of style libraries.

The following illustration shows the Sheet Metal Defaults dialog box. The first component is a sheet metal rule. A single sheet metal rule is used to drive the material, thickness, the unfolding rule, and the geometric options for the bends and corners.

Objectives

After completing this lesson, you will be able to:

- Explain how to use bend tables for unfolding rules.
- Explain how to manage and share standard sheet metal information.
- Describe sheet metal rules and how they affect geometry in a sheet metal part file.
- Identify the steps that are required to create sheet metal rules that control material, bends, corners, flat pattern creation, and the management of bend tables.
- Manage sheet metal rules with import and export options and save them to style libraries for use in any design.
Manage Bend Tables

You use bend tables to determine a flat pattern definition from your 3D folded model. Through years of analysis and experience, many sheet metal shops have calculated bend deduction values to use with specific materials and specific bend angles.

These bend deduction values are often incorporated into an Excel spread sheet file or an ASCI text file. Autodesk Inventor provides a user interface to manage bend tables.

The following illustration demonstrates a sheet metal unfold bend table that was imported from a text file.

Bend Table Pasted from Spreadsheet Image

When your bend deduction values are organized in a spreadsheet, you can copy them and then paste them into your bend table in Inventor.

Reference the Inventor sample folder for an example of how to format a spreadsheet file for copy and paste of bend deduction values.

In the following illustration, on the left, bend deduction values are highlighted in a spreadsheet, then copied to the clipboard. On the right, the values are pasted into the bend table for an unfold rule.
Import Text File

You can import a text file that contains bend deduction values. In the Styles and Standard Editor dialog box, with the Sheet Metal Unfold option activated, click Import. Navigate to the folder where your bend deduction values are stored and select the text file. Give the new unfold rule a name, and the data is imported.

Reference the Inventor sample folder for an example of how to format a text file for importing.

In the following illustration, a text file is selected for import to create a new bend table unfold rule.
Copy Bend Table

You can copy your bend table from Inventor and paste it into a spreadsheet or text file. Right-click in the root node of the bend table and select Copy Table to place the table on the clipboard.

You can also manage your column fit and paste a table from this shortcut menu.

The following illustration demonstrates the shortcut menu achieved when you right-click in the root cell of your table.

Bend Table Facts

- You can edit individual cells in your unfold rule bend table. Click any cell to activate it, then enter a new value.
- Unlike K-factors, which use bend allowance, a bend table uses bend deduction values.
- Set up a spread sheet to determine values based upon calculations and special formatting and then paste the calculated data into your Inventor bend table.

About Sheet Metal Defaults

Sheet metal defaults control the use of sheet metal rules. Sheet metal rules drive material styles and other geometric options of the sheet metal parts.

Sheet metal rules move the Inventor user away from using templates to maintain all sheet metal styles to using style libraries. This workflow more closely resembles the workflow of part modeling in Inventor. This results in one workflow regardless of the Inventor environment in which you work and decreases the size and complexity of your template files.

When migrating sheet metal parts from earlier releases, sheet metal styles are automatically converted to sheet metal rules.

The following illustration shows the Style and Standard Editor dialog box and demonstrates how a sheet metal rule encompasses information on the material, bends, and corner relief.
Sheet Metal Defaults Defined

Use sheet metal defaults to access sheet metal rules. As with other styles, you can also access the sheet metal rules through the Style and Standard Editor dialog box.

Use sheet metal defaults to activate a specific sheet metal rule or to quickly override a rule setting, such as thickness, material style, or an unfolding rule.

In the following illustration, the thickness value, set in the Default_mm sheet metal rule, is being overridden.
Example of Sheet Metal Defaults

When designing a sheet metal part, you specify what the material is, its thickness, and the other attributes that are used for calculating the flat pattern of the part.

At some point during the design project, you receive an engineering change. Based on part analysis and prototype testing, it is determined that the material needs to be galvanized and that the thickness of the material can decrease for 1.5 mm to 1.0 mm. Using sheet metal defaults from your predefined styles library, you select the 1.0 mm Galvanized style. All bend, corner, and unfold rules are already defined in the rule making the change simple and fast.

About Sheet Metal Rules

Sheet metal rules provide an efficient way to capture and reuse a standard set of properties and methods throughout all of your sheet metal designs.

In the following illustration, on the left, is a sheet metal part created using the default metric rule. On the right, the same part is shown using a different rule.

Sheet Metal Styles Rules

Sheet metal rules are a set of common properties that can be given a name and reused as needed. Using rules eliminates the need to make changes in every sheet metal design. You have may have many rules defined depending on the variety of work that you perform. Your rules can help you to efficiently set properties, such as material style, material thickness, and unfolding rules.

Similarity to Dimension Styles

Sheet Metal Rules are similar to Dimension styles in AutoCAD®. Sheet metal rules group material style and thickness and many other characteristics of the sheet metal part together so that they can be applied to a part in a single step. Similarly, a dimension style defines all aspects of the dimension appearance, such as text font and height, arrow style and size, and decimal precision.
Example of Sheet Metal Rules

You design a sheet metal part that will be made using 0.5 mm thick mild steel. For your 0.5 mm parts, you use the following values:

- Bend radius of two times the part thickness = 1.0 mm
- Relief width equal to the part thickness = 0.5 mm
- Relief depth equal to one-half the part thickness = 0.25 mm

An engineering analysis determines that the part should be made from 1.5 mm aluminum instead of mild steel. Aluminum requires different bend characteristics than mild steel. You need to change each of the values listed above, as well as the other values to make the change to your design.

Using sheet metal rules, you can assign a new rule to the part which has all of the values that are needed for a 1.5 mm thick aluminum part.

Creating and Using Sheet Metal Rules

Sheet metal rules are applied to the parts throughout the design process. Ideally, the data supplied by the rules is transparent to the designer, allowing the designer to concentrate on the design.

However, sheet metal rules must be created and added to the styles library so that they can be utilized when needed. Shops often work with a limited variety of materials and material thickness. Over time, each shop builds expertise using these materials and knows how a material will bend, fold, punch, and otherwise behave as operations are performed on it. This knowledge of materials is the essential information input into sheet metal rules, thereby capturing the expertise of the local enterprise and preserving it for future use.

After you create a sheet metal rule, you apply it to your sheet metal part using sheet metal defaults. All properties that are associated with the sheet metal rule are reflected in the current part.

In the following illustration, a new sheet metal rule is created which specifies 0.9 mm thick galvanized steel.
Access

Sheet Metal Rules

Ribbon: Sheet Metal tab > Setup panel

Toolbar: Sheet Metal Features > Sheet Metal Defaults

Style and Standard Editor Dialog Box

The following options are available in the Style and Standard Editor dialog box:

1. **Rule List**: Do the following to existing sheet metal rules:
   - Select a rule to modify it.
   - Right-click a rule to activate it, create a new rule based upon it, rename it, and export it.

2. **Unfold List**: Activate, edit, create, rename, and export sheet metal unfold rules.

3. **Back**: Returns to the previous selection.
Lesson: Sheet Metal Rules

New: Creates a new style based on the selected style.

Save: Saves the settings to the selected style.

Reset: Resets any unsaved changes to the currently selected rule.

Setting Tabs: Select a tab to change the settings associated with it.

Sheet Metal Rule: Sheet Tab

The following options are available in the Style and Standard Editor dialog box, Sheet Metal Rule, Sheet tab:

1. **Material**: Select a material type.
2. **Thickness**: Enter a material thickness.
3. **Unfolding Rule**: Select an unfold rule as defined in the Style and Standard Editor dialog box.
4. **Miter/Rip/Seam Gap**: Enter a value for the default gap size for Miter, Rip, or Seam features.
5. **Flat Pattern Bend Angle**: Select an option to report the bending angle or the open angle.
6. **Flat Pattern Punch Representation**: Select the punch representation to use when applying this rule. This is the initial representation; you can also override this setting.
Sheet Metal Rule: Bend Tab

The following options are available in the Style and Standard Editor, Sheet Metal Rule, Bend tab:

1. **Relief Shape**: Select a relief shape that is created for all features that create a bend in the part. Shape options include Tear, Round, and Straight.
2. **Relief Width**: Enter a value for the width of the bend relief.
3. **Relief Depth**: Enter a value for the depth of the bend relief.
4. **Minimum Remnant**: Enter a value representing the minimum stock that remains along side of the bend relief cut. If the remaining stock is less than the remnant size, it is removed.
5. **Bend Radius**: Enter a value for the inside bend radius.
6. **Bend Transition**: Select a bend transition that represents the bend geometry in the flat pattern. The options include None, Intersection, Straight Line, Arc, and Trim to Bend. In the folded model, a None transition exists for all types except a Trim transition.
Sheet Metal Rule: Corner Tab

The following options are available in the Style and Standard Editor, Sheet Metal Rule, Corner tab:

- **Two Bend Intersection Relief Shape**: Select a shape that defines the corner relief. Options include Round, Square, Tear, Trim to Bend, Linear Weld, and Arc Weld.
- **Two Bend Intersection Relief Size**: Enter a value for the size of the corner relief.
- **Three Bend Intersection Relief Shape**: Select a shape that defines the corner relief to display in the flat pattern. Options include No Replacement, Intersection, Full Round, and Round with Radius. The relief shape is not displayed in the folded model.
- **Three Bend Intersection Relief Radius**: Enter the default corner relief radius.

Create Sheet Metal Rules Described

You can create sheet metal rules by starting from the Sheet Metal Defaults dialog box, or directly from the Style and Standard Editor. With either method, the rule creation takes place in the Styles and Standard Editor dialog box.

Creating sheet metal rules is an alternative to using template based styles. With sheet metal rules, you can save and manage all of your material information in the styles library for consistency and reuse. A single sheet metal rule will control material style and thickness, sheet metal unfold rule, and bend corner and punch representation options.
In the following illustration, the Style and Standard Editor is shown with multiple sheet metals rules defined.

**Process: Creating Sheet Metal Rules**

The following steps give an overview of creating a sheet metal rule.

- In the Sheet Metal Defaults dialog box, click Edit Sheet Metal Rule.
- In the Style and Standard Editor, click New and enter a name for your new sheet metal rule.
- On the Sheet tab, assign a material and a material thickness.
- Select an Unfolding Rule and Flat Pattern Punch Representation.
- Make settings on the Bend and Corner tabs according to your design specifications.
- Click Save and Done to dismiss the Style and Standard Editor.
- In the Sheet Metal Defaults dialog box, set the new Rule active.
Managing Sheet Metal Rules

Sheet metal rules are created and managed with the Styles and Standards Editor dialog box in the same way in which you manage lighting, color, material, and annotation styles.

Sheet metal rules are created locally. You export a rule from your current part file to the styles library, thus making it available to you in all of your designs.

In the following illustration, you can save a sheet metal rule from the Style and Standard Editor dialog box to the current style library.

Access

Style and Standard Editor

Ribbon: Manage tab > Styles and Standards panel
**Style and Standard Editor Dialog Box**

The following options are available in the Style and Standard Editor dialog box.

1. Displays the sheet metal rules that are currently available in the active document or current style library depending on the filter settings.
2. Filter to display either All Styles or Local Styles.
3. Use Import to import a style definition file.

**Sheet Metal Rule Shortcut Options**

The following options are available in the shortcut menu of a sheet metal rule from the Style and Standard Editor dialog box.
Active: Use this option to make the selected rule active.

New Style: Use this option to create a new sheet metal rule based upon the selected rule.

Purge Style: Use this option to purge a local sheet metal rule from your current design.

Purge Style and Sub-Styles: Use this option to purge the selected rule and any substyles of that rule.

Export: Exports the selected rules to a style definition file (*.styxml).

Save to Style Library: Uses the Save Styles to Style Library dialog box to save the selected rules to the active style library.

Management of Your Sheet Metal Rules

You can create a style library for a specific project file that has all of your defined sheet metal rules and sheet metal unfold rules. This style library provides you with a single source to bring defined rules into your active part.

Process: Copying Sheet Metal Rules into the Active Style Library

The following steps outline the process of copying a sheet metal rule to a style library from the Style and Standard Editor dialog box.

Note: This process assumes that Use Style Library is set to Yes in the project file.

1. With a sheet metal part file open, create a new local sheet metal rule and save it.

2. In the Style and Standard Editor dialog box, right-click the new rule. Click Save to Style Library.
3. In the Save Styles to Style Library dialog box, under Save to Library, select Yes. The rule is written to the styles library.

4. With the filter set to Local Styles, right-click the new rule and purge it from the current file. Observe that it is no longer listed under the sheet metal rules.

5. Set the filter to All Styles. Right-click the new rule that you saved to the style library in a previous step. Click Active.

6. Set the filter back to Local Styles. Observe that the rule is once again listed under your available sheet metal rules.

Using Legacy Sheet Metal Styles

- When importing an Inventor file from a previous release, defined sheet metal styles convert to a sheet metal rule.
- You can save your sheet metal rules to your sheet metal template file, so that they are available in all new sheet metal parts that you create.
Exercise: Create a Sheet Metal Rule

In this exercise, you open an existing sheet metal part, create a new sheet metal rule, and apply the new rule to the part.

The completed exercise

Completing the Exercise
To complete the exercise, follow the steps in this book or in the onscreen exercise. In the onscreen list of chapters and exercises, click Chapter 1: Sheet Metal Overview. Click Exercise: Create and Save Sheet Metal Styles in a Template.

1. Open Sheet_Metal_Rules.ipt.
2. Activate the Sheet Metal Defaults tool.
3. In the Style and Standard Editor dialog box, click New.

Observe the current setting for the sheet metal rule, the material style, and the unfolding rule.
Notice that there is a thickness override.
Click Edit Sheet Metal Rule.
4. For the unfolding rule, select 18GA GAL_KFactor.

5. For the flat pattern punch representation, select Formed Punch Feature.
   - Click Save.
   - Click Done.

6. In the Sheet Metal Defaults dialog box, for the sheet metal rule, select 18GA Galvanized.
   - Select Use Thickness from Rule.
   - For the material style, select By Sheet Metal Rule (Galvanized Steel).
   - For the unfolding rule, select By Sheet Metal Rule (18GA GAL_KFactor).

7. Click OK. Observe the change in the thickness of the sheet metal part.
   **Tip:** Use the Undo and Redo tools to view the changes.
8. To import a style, activate the Style and Standard Editor:
   ■ Click Import.
   ■ In the Import Style Definition dialog box, select 18GA-Gold.styxml.
   ■ Click Open.
   ■ Click Done.
   Notice that the imported style is added to the list of sheet metal rules.

9. To activate the 18GA Gold sheet metal rule:
   ■ Activate the Sheet Metal Defaults tool.
   ■ For the sheet metal rule, select 18GA Gold.
   ■ Click OK.

10. Click Application menu > Save As:
    ■ In the Save as dialog box, for the file name, enter Sheet_Metal_Rules2.ipt.
    ■ Click Save.
    ■ Close Sheet_Metal_Rules2.ipt.

11. Click Get Started tab > Launch panel > Projects.
    ■ In the lower part of the Projects dialog box, right-click Use Style Library = Read Only.
    ■ Click Yes.
    ■ Click Save.
    ■ Click Done.

    Note: If you did not create Sheet_Metal_Rules2.ipt, open Sheet_Metal_Rules_Gold.ipt.

13. Observe the local styles:
    ■ Activate the Style and Standard Editor.
    ■ In the upper right corner of the dialog box, for the Styles Filter, select Local Styles.
    ■ Expand Sheet Metal Rule.
    ■ Notice that there are four sheet metal rules in the local part file.
14. To save the new style to the style library:
   ▪ Under Sheet Metal Rule, right-click 18GA Galvanized.
   ▪ Click Save to Style Library.
   ▪ In the Save Styles to Style Library dialog box, click OK.

15. Close the Style and Standard Editor and the Sheet_Metal_Rules2.ipt file.


17. Activate the Style and Standard Editor.
   ▪ Expand Sheet Metal Rule.
   ▪ Observe that only the Default rule exists locally in the part file.

18. For the styles filter, select All Styles.
   ▪ Observe the available sheet metal rules.
   ▪ Under Sheet Metal Rules, right-click 18GA Galvanized.
   ▪ Click Active.
   ▪ For the filter styles, select Local Styles.
   **Note:** When a style is made active from the style library, it is copied to the local file that is being modified.

19. Close the Style and Standard Editor.

20. Close all files. Do not save.
Chapter Summary

By applying knowledge of the foundational elements of sheet metal design including terminology, equipment, and sheet metal rules, you can learn to create simple and complex sheet metal parts efficiently.

Having completed this chapter, you can:

- Describe common sheet metal concepts, terms, and manufacturing equipment, and use Autodesk Inventor to design a simple sheet metal part.
- Describe the two methods for designing sheet metal parts and use them to create sheet metal designs in Autodesk Inventor.
- Create sheet metal rules incorporating style and materials information for common sheet metal parts.