Creo Parametric 3.0: Sheet Metal Design

Student Guide
1st Edition
ASCENT - Center for Technical Knowledge®
Creo Parametric 3.0
Sheet Metal Design
1st Edition

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Preface

The Creo Parametric 3.0: Sheet Metal Design student guide enables you to use your introductory modeling skills to create sheet metal models, including wall, bends, notches, and form features. On completion of this course, you will have acquired the skills to confidently manipulate sheet metal geometry, adjust bend developed lengths, and convert solid parts.

Topics Covered:

• The sheet metal environment
• Primary and secondary walls
• Bend relief
• Corner relief
• Regular unbends, back bends, and cuts
• Notches and punches
• Bend features
• Unbending complex geometry
• Sheet metal forms
• Documenting a sheet metal part
• Converting solid parts
• Sheet metal setup
• Investigating a sheet metal part
Note on Software Setup

This student guide assumes a standard installation of the software using the default preferences during installation. Lectures and practices use the standard software templates and default options for the Content Libraries.

This guide was developed against Creo Parametric 3.0 Build M070.

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Scott Hendren has been a trainer and curriculum developer in the PLM industry for almost 20 years, with experience on multiple CAD systems, including Pro/ENGINEER, Creo Parametric, and CATIA. Trained in Instructional Design, Scott uses his skills to develop instructor-led and web-based training products.

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In this Guide

The following images highlight some of the features that can be found in this Student Guide.

Practice Files

The Practice Files page tells you how to download and install the practice files that are provided with this student guide.

FTP link for practice files

Chapters

Each chapter begins with a brief introduction and a list of the chapter’s Learning Objectives.

Learning Objectives for the chapter

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1.3 Working with Commands

Starting Commands

The main way to access commands in the AutoCAD software is to use the Ribbon. Several of the file commands are available in the Quick Access Toolbar, and some other commands are available in the Status Bar or through shortcut menus. There are additional access methods, such as Tool Palettes. The names of all of the commands can also be typed in the Command Line. A table is included to help you identify the various methods of accessing the commands.

When typing the name of a command in the Command Line or on the AutoCAD Task Pane, the command is automatically completed for you. For example, the AutoScale command is automatically completed for you. It also supports multiple options by displaying all of the commands that contain the word that you typed, as shown in Figure 1-12. You can then scroll through the list and select a command.

### Instructional Content

Each chapter is split into a series of sections of instructional content on specific topics. These lectures include the descriptions, step-by-step procedures, figures, hints, and information you need to achieve the chapter's Learning Objectives.

### Practice Objectives

**Practice 1c**

Saving a Drawing File

Estimated time for completion: under 5 minutes.

In this practice you will open a drawing, save it, and modify the Automatic Save Options, as shown in Figure 1-31.

1. Open Building Valley.1.dwg from your class files folder.
2. Click Quick Access Toolbar, click [Save]. In the Command Line, Enter [Save] displays indicating that the AutoCAD software has performed a quick save.
3. In the Application Menu, click Options dialog box.
4. In the Open and Save tab, change the time for Automatic save to 15 minutes.

### Practices

Practices enable you to use the software to perform a hands-on review of a topic.

Some practices require you to use prepared practice files, which can be downloaded from the link found on the Practice Files page.

### Chapter Review Questions

Chapter review questions, located at the end of each chapter, enable you to review the key concepts and learning objectives of the chapter.
Command Summary

The Command Summary is located at the end of each chapter. It contains a list of the software commands that are used throughout the chapter, and provides information on where the command is found in the software.
Chapter 1

Introduction to Sheet Metal Modeling

The Creo Parametric sheetmetal tools enables you to create thin walled parts that can be bent and flattened to form the final geometry. This chapter provides an overview of the sheetmetal environment and workflow.

Learning Objectives in this Chapter

- Understand the benefits of using the Creo Parametric Sheet Metal environment to design sheet metal models.
- Identify the similarities in generating a sheet metal part and a solid part.
- Identify the color scheme when creating a sheet metal part in Creo Parametric.
- Identify parameters that can be specified when a sheet metal part is created.
- Understand how sheet metal parts are oriented using the commands that are also used in part mode.
- Gain a general understanding of the formula that Creo Parametric uses to compensate for the stretching and compression that occurs in areas that are bent.
- Understand the overall workflow that is used to create a sheet metal model using feature based and top-down design techniques.
- Identify the method that can be used to convert a solid part into a sheet metal part.
1.1 Sheet Metal Environment

Working in Sheetmetal mode to design sheet metal parts enables you to efficiently and effectively capture design intent. Sheetmetal mode also enables you to design components for the following uses:

- Design a model within the context of an assembly so that all 3D information is present.
- Create features specific to the sheet metal modeling process.
- Create different instances of the model to use at different times, such as for design and documentation.
- Extract information and establish controls that are beneficial to the manufacturing process.
- Generate reports and other information to document the design of the sheet metal part.

The process of creating a sheet metal model is similar to the process of creating a solid part in the following ways:

- Individual features (e.g., walls, bends, notches, bend back, and forms) are created in sequence and reference one another, resulting in parent/child relationships.
- Sketching references are established when creating certain types of walls, just as they are for protrusions and cuts.
- Depth options are specified in Sheetmetal mode.

Parent/child references and design intent are equally important in Sheetmetal mode and solid part creation. You must carefully consider feature type and order of creation.
1.2 Display of Sheet Metal Parts

Geometry created in Sheetmetal mode has a specific color scheme; one side of the model is green and the other side is white, as shown in Figure 1–1.

When modeling a sheet metal part, remember that it always has a constant wall thickness. The green side of the model is used as the driving side. The white surface is then offset by a distance that is equal to the material thickness. Typically, references and geometry are generated based on the green side of the model.
1.3 Parameters

A default material thickness can be set for a sheet metal part. Once a sheet metal part has been created, select **Model Intent>Parameters**. The Parameters dialog box lists the **SMT_THICKNESS** parameter, as shown in Figure 1–2. When you create sheet metal features they use this value by default.

![Figure 1–2](image)

When the first wall is created, the parameter value entered in the Parameters dialog box is automatically assigned in the **Thickness** field in the dashboard, as shown in Figure 1–3.

![Figure 1–3](image)
1.4 Orienting a Sheet Metal Part

Models are oriented in Sheetmetal mode similar to how parts are oriented in Part mode. You can pan, zoom, and spin a model using the same options. You can also display the model in a 2D orientation using the Orientation dialog box (Reorient) and selecting two planar surfaces to face Front, Back, Top, Bottom, Left, or Right. When displaying a 2D orientation in Sheetmetal mode, the first viewing reference selected must be a planar face or datum plane, as shown in Figure 1–4. The second reference can be a planar surface or an edge.

Use (Pick From List) (shortcut menu) when selecting edges and surfaces for orientation to ensure that the intended reference is selected.

![Figure 1–4](Bottom (select edge))
1.5 Developed Length of Sheet Metal Bends

One of the primary methods for manipulating sheet metal is bending (forming). Stretching and compression occur in some areas when the part is bent. As a result, material thickness on the part can vary.

Creo Parametric compensates for this by using a formula that considers material thickness, bend radius, bend angle, and other properties. This formula locates a neutral bend line and measures its length to determine the developed length of the bend. With this technique, Sheetmetal mode captures your design intent when creating the model in the bent (formed) condition. You can then create a flat instance of the model to be used for manufacturing, as shown in Figure 1–5.

The calculation of the developed length can also be defined in a bend table. A bend table includes information about the bend radius, sequence, and angle.
1.6 Designing in Sheetmetal Mode

A common way of creating sheet metal parts in Creo Parametric is to build them using feature-based techniques, similar to those used when building a solid part.

How To: Create a Part in Sheetmetal Mode

1. Open a template or create default datum planes in a new model, as shown in Figure 1–6.

   ![Figure 1–6](image)

2. Create any required reference geometry.
3. Create the primary wall feature, as shown in Figure 1–7.

   ![Figure 1–7](image)
4. Create additional walls, as shown in Figure 1–8.

5. Apply a bend relief, as shown in Figure 1–9.

6. Create notches, punches, and cuts, as shown in Figure 1–10.
7. Create forms as shown in Figure 1–11. In this example they are used to stiffen the opening in the part.

Figure 1–11

Form added to rear wall

8. Create a flat state to facilitate documentation, as shown in Figure 1–12.

Figure 1–12

9. Create a detail drawing including bend information to document the design, as shown in Figure 1–13.

Figure 1–13
Assembly Mode

If you use top-down design principles, working with an assembly could be more beneficial. Sheet metal parts created within the context of an assembly model often act as supporting structures for principle components in a larger assembly.

How To: Create a Sheet Metal Part in Assembly Mode

1. Assemble all of the required components relative to each other or to a default reference.
2. Create the required sheet metal components with appropriate parent/child references.
3. Add any remaining components.
4. Apply an appropriate bend table to the sheet metal part so that it has correct clearance with the surrounding components.
5. Define a bend order table in Sheetmetal mode to generate the bend sequence for the part.
6. Create a flat state to display fully formed and flattened instances of the part on a drawing.
7. Create a detail drawing, including bend information.

Convert Parts

You can also create a sheet metal model by converting a solid part.

How To: Convert a Solid Part to Sheet Metal

1. Open the part and select Operations>Convert to Sheetmetal in the Model tab. Based on the type of part, click (Driving Surface) or (Shell). If the model is a uniform thickness, click (Driving Surface) and if it is non-uniform, click (Shell).
2. Add relief as required.
3. Add any required additional features.

Figure 1–14 shows an example of a converted part.

The top, front, and bottom surfaces were removed with a shell feature to achieve a constant wall thickness

Figure 1–14