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Creo Parametric 7.0
Cable and Harness Design

Learning Guide

1st Edition

ASCENT - Center for Technical Knowledge®
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Cable and Harness Design
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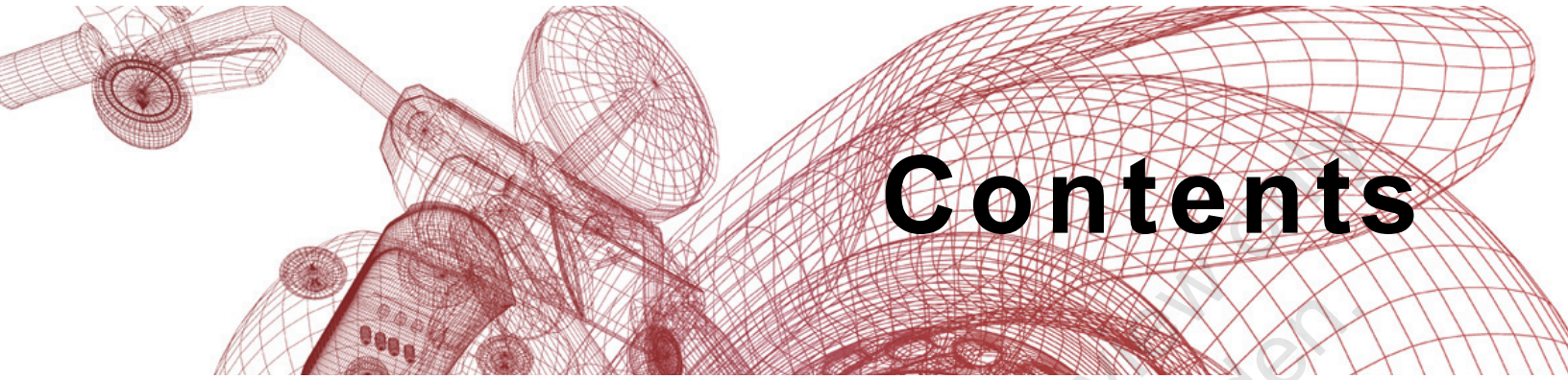

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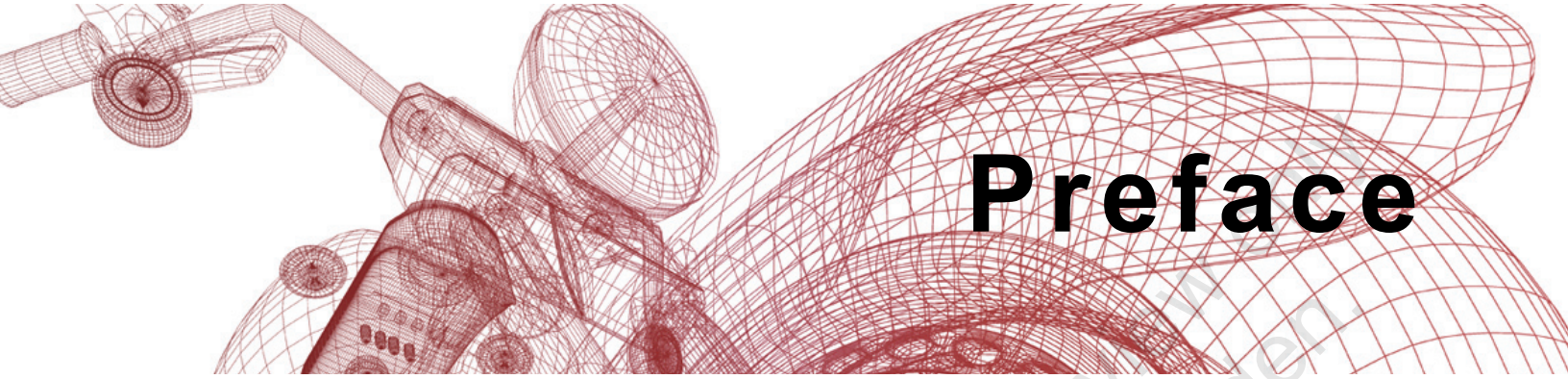


Preface

As an experienced user in the basics of Creo Parametric 7.0, this learning guide enables you to create electromechanical cabling systems designed in Creo Parametric using the Piping and Cabling Extension. Utilizing the parametric and associative nature of Creo Parametric, an electromechanical designer can easily create realistic 3D cabling assemblies, wire lists, bill of material tables, and nail-board drawings.

The *Creo Parametric 7.0: Cable and Harness Design* learning guide contains numerous practices to give you practical experience that will improve your job performance.

Topics Covered

- Cabling Process Overview
- Cabling Terminology
- Environment and Configuration Setup
- Electromechanical Model Setup
- Manual Designation and Parameters
- Manual Spools
- Manual Cabling Features
- Logical Reference Technique
- Routing Methods
- Modifying Cabling Assemblies
- Additional Routing Features
- Networking
- Cabling Assembly Deliverables
- HARNESS-MFG

Prerequisites

- Access to the Creo Parametric 7.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included with this guide are compatible with the commercial version of the software, but not the student edition.
- It is recommended that you have completed the C*reo Parametric 7.0: Introduction to Solid Modeling* learning guide, or have equivalent experience.

Note on Software Setup

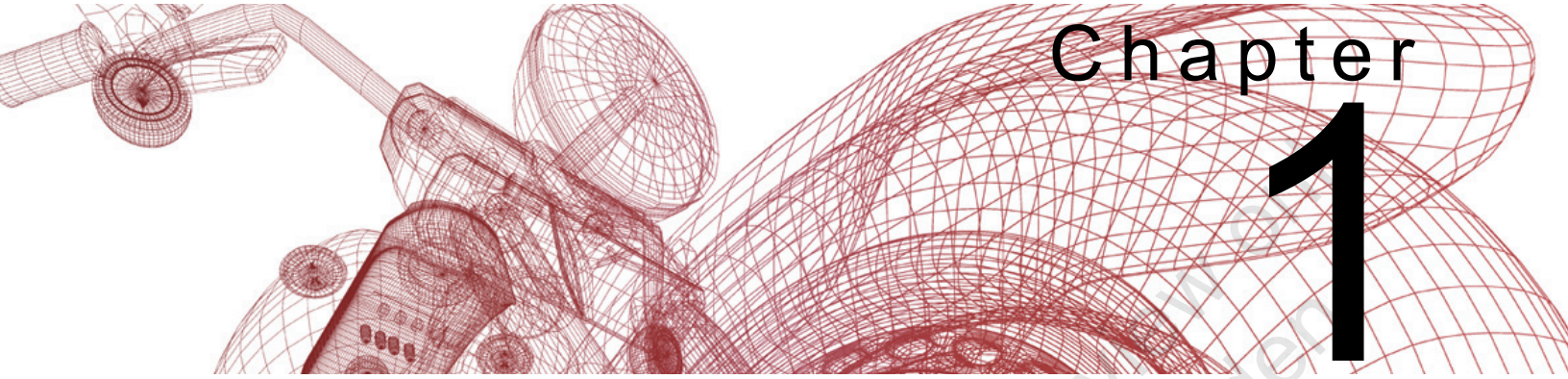
This guide assumes a standard installation of the software using the default preferences during installation. Lectures and practices use the standard software templates and default options for the Content Libraries.

This content was developed using Creo Parametric 7.0, Build 7.0.2.0.

In This Guide

The following highlights the key features of this guide.

Feature	Description
Practice Files	The Practice Files page includes a link to the practice files and instructions on how to download and install them. The practice files are required to complete the practices in this guide.
Chapters	A chapter consists of the following - Learning Objectives, Instructional Content, Practices, Chapter Review Questions, and Command Summary. • **Learning Objectives** define the skills you can acquire by learning the content provided in the chapter. • **Instructional Content**, which begins right after Learning Objectives, refers to the descriptive and procedural information related to various topics. Each main topic introduces a product feature, discusses various aspects of that feature, and provides step-by-step procedures on how to use that feature. Where relevant, examples, figures, helpful hints, and notes are provided. • **Practice** for a topic follows the instructional content. Practices enable you to use the software to perform a hands-on review of a topic. It is required that you download the practice files (using the link found on the Practice Files page) prior to starting the first practice. • **Chapter Review Questions**, located close to the end of a chapter, enable you to test your knowledge of the key concepts discussed in the chapter.



Chapter 1

Process Overview

You can design electromechanical cabling systems in Creo Parametric using the Creo Schematics, Piping and Cabling Extension, and Harness-MFG modules. Pro/REPORT can be used for design documentation. Using the parametric and associative characteristics of Creo Parametric, an electromechanical designer can easily create 2D schematics, realistic 3D cabling assemblies, wire lists, bill of material tables, and nail-board drawings.

Learning Objective in This Chapter

- Understand the Electromechanical design process.

1.1 Electromechanical Overview

A process overview of the electromechanical cabling design process using Creo Parametric is shown in Figure 1–1.

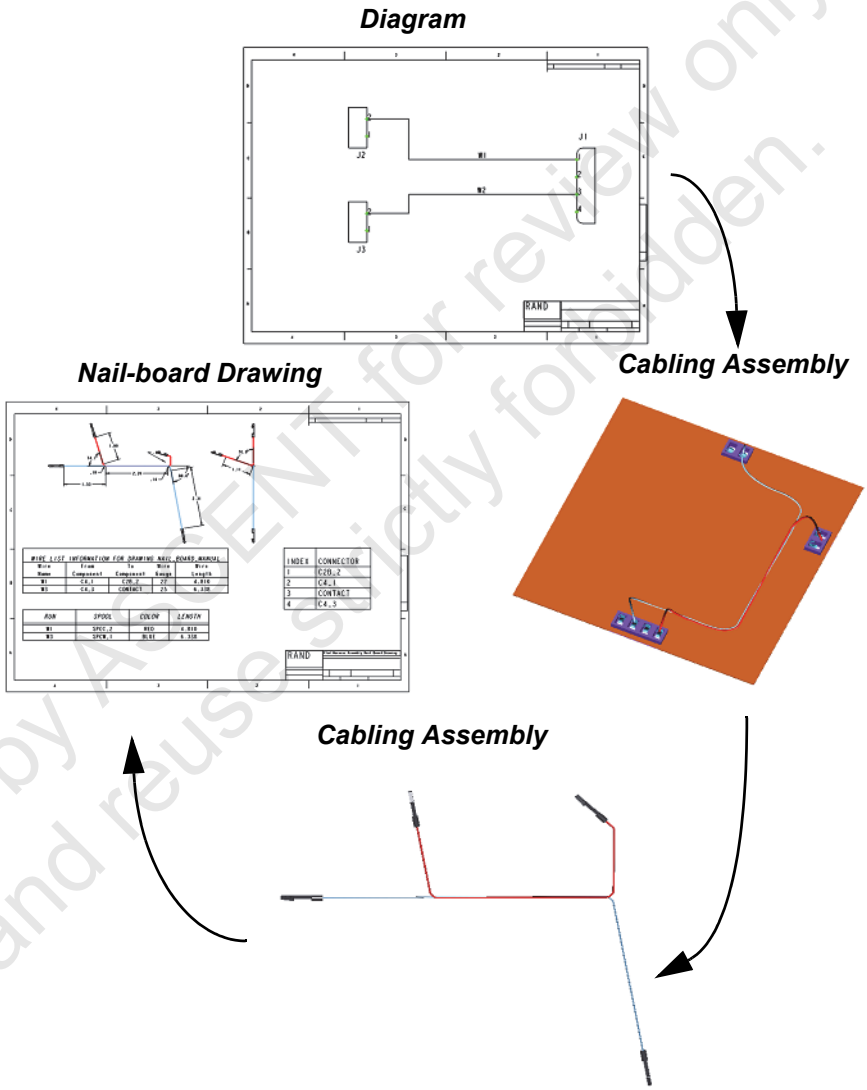


Figure 1–1

This overview is provided as a reference to use throughout this learning guide. Each stage of the process is discussed in detail in related chapters.

The Creo Schematics course covers the creation of a 2D schematic to define the wire connection information. The Cabling and Harness-MFG course covers the creation of the 3D cabling assembly, flattening of the 3D wiring harness, and manufacturing documentation.

Process Overview

1. Decide which cabling entities will be routed to various electromechanical devices in the assembly.
2. Using Creo Schematics, create a 2D schematic to represent the logical connectivity between cabling entities and their respective electromechanical devices.
3. Using Cabling, create a realistic 3D assembly by routing cable geometry throughout the assembly. Logical connectivity and parameters that are defined in Creo Schematics can be used in Cabling to expedite the design process.
4. Using HARNESS-MFG, create a flattened harness to simulate laying out the wire and cable harness geometry on a nail-board.
5. Create a 2D drawing to document the flattened harness for manufacturing. Use Pro/REPORT to create a customized bill of materials and wire list. Drawings can also be created for the routed 3D assemblies.

The process outlined above describes how Creo Parametric is intended to be used to design electromechanical cabling systems. Sometimes it is not practical to create a 2D schematic using Creo Schematics as part of the electromechanical cable design process.

Alternative methods of creating a 3D assembly without linking to a 2D schematic are covered in this learning guide.