Sheetmetal using Creo Parametric 3.0

Overview

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Course Code	WBT-4507-0
Course Length	16 Hours

In this course, you will learn how to create sheetmetal parts in Creo Parametric. The course builds upon the basic lessons you learned in Introduction to Creo Parametric 3.0 and serves as the second stage of learning. In this course, you will learn how to design sheetmetal parts and assemblies, including sheetmetal production drawings. All the functions needed to create sheetmetal parts, drawings, and assemblies are covered. Upon completion of this course, you will be able to create sheetmetal design models, create the flat state of the model, and document both in production drawings.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.
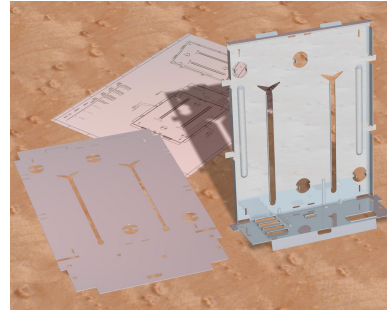
Course Objectives

- Create, convert, and display the sheetmetal model
- Use methods of developed length calculation
- Use primary and secondary wall features, as well as partial walls
- Use bend relief
- Use unbend and bend back features
- Apply sheetmetal bend features
- Use flat patterns
- Create sheetmetal cuts
- Create forms
- Use notch and punch features
- Utilize the sheetmetal environment setup, sheetmetal design information tools, and sheetmetal design rules
- Detail sheetmetal designs
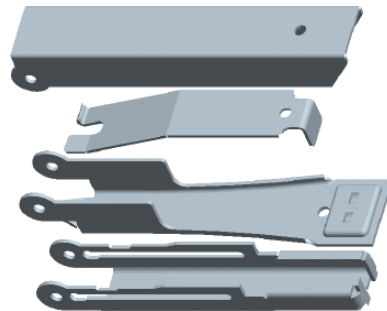
Prerequisites

- Introduction to Creo Parametric 3.0

Audience

- This course is intended for design engineers, mechanical designers, and industrial designers. People in related roles can also benefit from taking this course.

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