



Creo Parametric 3.0 Basics

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If you like to learn by doing, then this book shows you to design and document mechanical components one-step at a time. Brief explanations followed by step-by-step instructions, and a real-world example make it a good starting point to learn Creo Parametric.

The author introduces the user-interface, and then starts solid modeling. You will learn to draw sketches, create 3D parts and assemble them, create drawings, sheet metal parts, and design complex shapes using surface modeling tools. The examples in this book help you to discover the use of various tools and capabilities of Creo Parametric. Following are some of the skills you acquire after completing this book:

- A good command over the user-interface
- Create and edit parametric sketches
- Create basic and complex solid models
- Create assemblies using Bottom-up and Top-down approaches
- Create manufacturing drawings
- Sheet metal design
- Create and edit splines
- Create complex and ergonomic surfaces

Creo Parametric 3.0 Basics is a self-study guide. You can learn various tools or work with examples. In both ways, you can develop a good foundation in solid modeling.

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