



# CATIA V5-6R2014 For Beginners

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# CATIA V5-6R2014 For Beginners

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## **CATIA V5-6R2014 For Beginners** CADFolks

This book helps you to get started with CATIA V5 using step-by-step examples. It starts with creating sketches and parts, assembling them, and then creating print ready drawings. This book gives you an idea about how you can design and document various mechanical components, and helps you to learn some advanced tools and techniques. This book follows some of the best practices in creating parts. In addition to this, there are additional chapters covering sheet metal and surface design. Each topic in this has a brief introduction and a step-by-step example. This will help you to learn CATIA V5 quickly and easily.

- Familiarize yourself with the User Interface
- Learn some best practices to create sketches and 3D components
- Learn additional part modelling tools
- Learn to create Multi-body parts
- Learn to modify components keeping in mind the design intent
- Teach yourself to create assemblies
- Learn Top-down assembly design
- Learn to create 2D drawings
- Create basic sheet metal parts
- Create sheet metal drawings
- Create complex shapes using surface modeling tools

Downloadable tutorial and exercise file from the companion website.

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