



Parametric Modeling with Autodesk Inventor 2017

Randy Shih

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Parametric Modeling with Autodesk Inventor 2017 contains a series of sixteen tutorial style lessons designed to introduce Autodesk Inventor, solid modeling, and parametric modeling. It uses a hands-on, exercise-intensive approach to all the important parametric modeling techniques and concepts.

The lessons guide the user from constructing basic shapes to building intelligent mechanical designs, creating multi-view drawings and assembly models. Other featured topics include sheet metal design, motion analysis, 2D design reuse, collision and contact, stress analysis and the Autodesk Inventor 2017 Certified User Examination.

Table of Contents

1. Getting Started
 2. Parametric Modeling Fundamentals
 3. Constructive Solid Geometry Concepts
 4. Model History Tree
 5. Parametric Constraints Fundamentals
 6. Geometric Construction Tools
 7. Parent/Child Relationships and the BORN Technique
 8. Part Drawings and Associative Functionality
 9. Datum Features and Auxiliary Views
 10. Symmetrical Features in Designs
 11. Advanced 3D Construction Tools
 12. Sheet Metal Designs
 13. Assembly Modeling - Putting It All Together
 14. Content Center and Basic Motion Analysis
 15. 2D Design Reuse, Collision and Contact
 16. Introduction to Stress Analysis
- Appendix
Index

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