



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.

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