

Catia V5-6r2015: Introduction to Modeling (Paperback)

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Reviews

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CATIA V5-6R2015: INTRODUCTION TO MODELING (PAPERBACK)



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Ascent, Center for Technical Knowledge, United States, 2017. Paperback. Condition: New. Language: English . Brand New Book ***** Print on Demand *****. The CATIA V5-6R2015: Introduction for NC and FEA Engineers student guide, you learn the process of designing models with CATIA V5 from conceptual sketching, through to solid modeling, assembly design, and drawing production. Upon completion of this student guide, you will have acquired the skills to confidently work with CATIA V5. Gain an understanding of the parametric design philosophy of CATIA V5 in this extensive hands-on student guide. It is expected that all new users of CATIA V5 need to complete this student guide. Topics Covered Overview of Parametric Design Process Customization of CATIA V5 Environment Creating and Constraining Sketch Geometry Sketched Feature Techniques and Formulas Adding Material with Pad and Shaft Features Removing Material with Pocket and Groove Features Creating Reference Elements for construction and measurement Fillet, Chamfer, Hole, Draft, and Shell Dress-Up Features Pattern, Copy, and Mirror Duplication Features Thin Features Nulti-sections Solid Features Feature Management Using the Hide / Show, Activate / Deactivate Functions Parent/Child Relationships and Feature Resolution Assembly Design Workbench Constraint creation, assembly management, and PDM considerations Obtaining Assembly Information (Measure, Clash, and Bill of Materials) Standard Parts from Catalogues and Save Management Working with Multi-Body Models Effective Modeling Tips and Techniques Prerequisites Experience in mechanical design and drawing production is recommended.

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