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CATIA V5-6R2020 for Designers, 18th Edition Prof. Sham Tickoo, 2021-01-19 CATIA V5 6R2020 for Designers is a comprehensive book written with the intention of helping the readers effectively use all solid modeling tools and other features of CATIA V5 6R2020 This book provides elaborative and clear explanation of the tools of all commonly used workbenches of CATIA V5 6R2020 After reading this book you will be able to create assemble and draft models The chapter on the DMU Kinematics workbench will enable the users to create edit simulate and analyze different mechanisms dynamically The chapter on the FreeStyle workbench will enable the users to dynamically design and manipulate surfaces The book explains the concepts through real world examples and the tutorials used in this book ensure that the users can relate the knowledge gained from this book with the actual mechanical industry designs Salient Features Consists of 19 chapters that are organized in a pedagogical sequence Tutorial approach to explain the concepts of CATIA V5 6R2020 Detailed explanation of CATIA V5 6R2020 tools First page summarizes the topics covered in the chapter Step by step instructions that guide the users through the learning process More than 40 real world mechanical engineering designs as tutorials and projects Additional information is provided throughout the book in the form of notes and tips Self Evaluation Tests and Review Questions provided at the end of each chapter to help users assess their knowledge Table of Contents Chapter 1 Introduction to CATIA V5 6R2020 Chapter 2 Drawing Sketches in the Sketcher Workbench I Chapter 3 Drawing Sketches in the Sketcher Workbench II Chapter 4 Constraining Sketches and Creating Base Features Chapter 5 Reference Elements and Sketch Based Features Chapter 6 Creating Dress Up and Hole Features Chapter 7 Editing Features Chapter 8 Transformation Features and Advanced Modeling Tools I Chapter 9 Advanced Modeling Tools II Chapter 10 Working with the Wireframe and Surface Design Workbench Chapter 11 Editing and Modifying Surfaces Chapter 12 Assembly Modeling Chapter 13 Working with the Drafting Workbench I Chapter 14 Working with the Drafting Workbench II Chapter 15 Working with Sheet Metal Components Chapter 16 DMU Kinematics Chapter 17 Introduction to Generative Shape Design Chapter 18 Working with the FreeStyle Workbench Chapter 19 Introduction to FEA and Generative Structural Analysis Student Projects Index

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Creo Parametric 7.0 for Designers, 7th Edition Prof. Sham Tickoo, 2020-11-29

Creo Parametric 7 0 for Designers book is written to help the readers effectively use the modeling and assembly tools by utilizing the parametric approach of Creo Parametric 7 0 effectively This book provides detailed description of the tools that are commonly used in modeling assembly sheetmetal as well as in mold design This book also covers the latest surfacing techniques like Freestyle and Style with the help of relevant examples and illustrations The Creo Parametric 7 0 for Designers book further elaborates on the procedure of generating the drawings of a model or assembly which are used for documentation of a model or assembly It also includes the concept of Geometric Dimensioning and tolerancing The examples and tutorials are used in this book to ensure that the users can relate the knowledge of this book with the actual mechanical industry designs Every chapter begins with a tools section that provides a brief information of the Creo Parametric tools This approach allows the user to use this book initially as a learning tool and then as a reference material Salient Features Consists of 17 chapters with comprehensive coverage of all concepts and techniques Tutorial approach to explain the concepts Detailed explanation of all commands and tools Summarized content on the first page of the topics that are covered in the chapter Hundreds of illustrations and step by step instructions for easy understanding Real world mechanical engineering designs as tutorials and exercises Additional projects for practice Additional information throughout the book in the form of notes and tips Self Evaluation Tests and Review Questions at the end of the chapters to help the users assess their knowledge Table of Contents Chapter 1 Introduction to Creo Parametric 7 0 Chapter 2 Creating Sketches in the Sketch Mode I Chapter 3 Creating Sketches in the Sketch Mode II Chapter 4 Creating Base Features Chapter 5 Datums Chapter 6 Options Aiding Construction of Parts I Chapter 7 Options Aiding Construction of Parts II Chapter 8 Options Aiding Construction of Parts III Chapter 9 Advanced Modeling Tools Chapter 10 Assembly Modeling Chapter 11

Generating Editing and Modifying the Drawing Views Chapter 12 Dimensioning the Drawing Views Chapter 13 Other Drawing Options Chapter 14 Working with Sheetmetal Components Chapter 15 Surface Modeling Chapter 16 Introduction to Mold Design Chapter 17 Concepts of Geometric Dimensioning and Tolerancing Index For free download from www.cadcim.com Siemens NX 2019 for Designers, 12th Edition Prof. Sham Tickoo,2019 Siemens NX 2019 for Designers is a comprehensive book that introduces the users to feature based 3D parametric solid modeling using the NX software The book covers all major environments of NX with a thorough explanation of all tools options and their applications to create real world products In this book about 40 mechanical engineering industry examples are used as tutorials and an additional 35 as exercises to ensure that the users can relate their knowledge and understand the design techniques used in the industry to design a product After reading the book the user will be able to create parts assemblies drawing views with bill of materials and learn the editing techniques that are essential to make a successful design Also in this book the author emphasizes on the solid modeling techniques that improve the productivity and efficiency of the user Keeping in mind the requirements of the users the book at first introduces sketching and part modeling in NX and then gradually progresses to cover assembly surfacing and drafting To make the users understand the concepts of Mold Design a chapter on mold designing of the plastic components is available in the book In addition a new chapter on basic concepts of GD T has also been added in this book Both these chapters are available for free download Written with the tutorial point of view and the learn by doing theme the book caters to the needs of both novice and advanced users of NX and is ideally suited for learning at your convenience and pace Salient Features Comprehensive coverage of NX concepts and techniques Tutorial approach to explain the concepts and tools of NX Detailed explanation of all commands and tools Hundreds of illustrations for easy understanding of concepts Step by step instructions to guide the users through the learning process More than 40 real world mechanical engineering designs as tutorials 35 as exercises and projects with step by step explanation Additional information throughout the book in the form of notes and tips Self Evaluation Tests and Review Questions at the end of each chapter to help the users assess their knowledge Table of Contents Chapter 1 Introduction to NX Chapter 2 Drawing Sketches for Solid Models Chapter 3 Adding Geometric and Dimensional Constraints to Sketches Chapter 4 Editing Extruding and Revolving Sketches Chapter 5 Working with Datum Planes Coordinate Systems and Datum Axes Chapter 6 Advanced Modeling Tools I Chapter 7 Advanced Modeling Tools II Chapter 8 Assembly Modeling I Chapter 9 Assembly Modeling II Chapter 10 Surface Modeling Chapter 11 Advanced Surface Modeling Chapter 12 Generating Editing and Dimensioning the Drawing Views Chapter 13 Synchronous Modeling Chapter 14 Sheet Metal Design Chapter 15 Introduction to Injection Mold Design For Free Download Chapter 16 Concepts of Geometric Dimensioning and Tolerancing For Free Download Index **Exploring Oracle Primavera P6 R8.4** Prof. Sham Tickoo,2017-01-11 Exploring Oracle Primavera P6 R8 4 book explains the concepts and principles of project management through practical examples tutorials and exercises This enables the users to harness

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