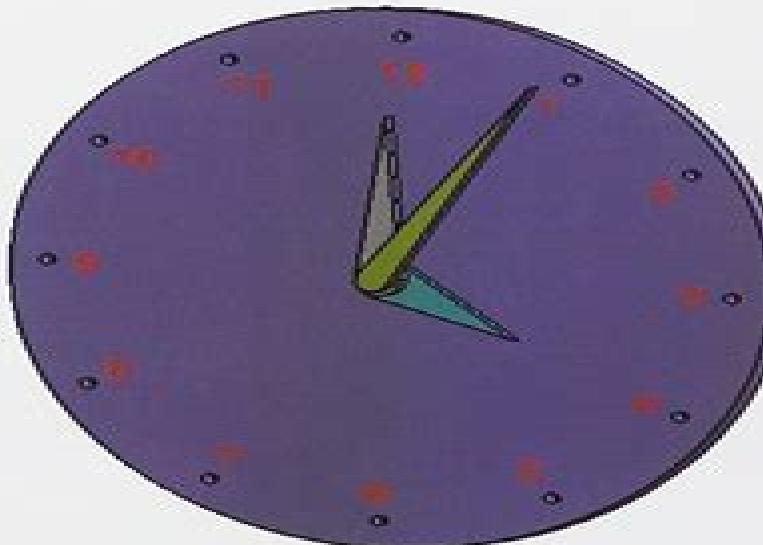


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Creating Dress Up and Hole Features Chapter 7 Editing Features Chapter 8 Transformation Features and Advanced Modeling Tools I Chapter 9 Advanced Modeling Tools II Chapter 10 Working with the Wireframe and Surface Design Workbench Chapter 11 Editing 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 Index *Catia V5-6r2017 Basics* Tutorial Books, 2017-09-10 CATIA V5 6R2017 Basics introduces you to the CATIA V5 user interface basic tools and modeling techniques. It gives users a strong foundation of CATIA V5 and covers the creation of parts assemblies drawings sheetmetal parts and complex shapes. This textbook helps you to know the use of various tools and commands of CATIA V5 as well as learn the design techniques. Every topic of this textbook starts with a brief explanation followed by a step by step procedure. In addition to that there are tutorials, exercises and self test questionnaires at the end of each chapter. These ensure that the user gains practical knowledge of each chapter before moving on to more advanced chapters. Table of Contents 1 Getting Started with CATIA V5 6R2017 2 Sketcher Workbench 3 Basic Sketch Based Features 4 Holes and Dress Up Features 5 Patterned Geometry 6 Rib Features 7 Multi Section Solids 8 Additional Features and Multibody Parts 9 Modifying Parts 10 Assemblies 11 Drawings 12 Sheet Metal Design 13 Surface Design. If you are an educator you can request an evaluation copy by sending us an email to online books999@gmail.com

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