

Catia V5 Workbook Release V5 6r2013

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2.2 CATIA V5 Workbook. CATIA V5 Standard Screen Layout . The following standard screen layout shows you where different tools and toolbars are located. The numbers coordinate with the following pages where the tool label is bolded. The tool label is followed by a brief explanation and in some cases, steps on how to use and/or access the tool.

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Book Description This workbook is an introduction to the main Workbench functions CATIA V5 has to offer. The book's objective is to instruct anyone who wants to learn CATIA V5 through organized, graphically rich, step-by-step instructions on the software's basic processes and tools. This book is not intended to be a reference guide.

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☐• Designed for users completely new to CATIA • Presents the workbenches, toolbars and tools using real life design problems • Uses organized, graphically rich, step-by-step instructions • This edition has been expanded to cover several additional workbenches ...

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CATIA V5 then takes the information and updates the Joggled Extrusion.CATPart according to the supplied input. CATIA V5 also automatically updates the standard dimensioned drawing (CATDrawing). The dimensioned drawing is ready to be released to the production floor in a matter of minutes instead of 30 to 60 minutes.

1-58503-321-9 -- Advanced CATIA V5 Workbook (Release 16)

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This workbook is an introduction to the main Workbench functions CATIA V5 has to offer. The book's objective is to instruct anyone who wants to learn CATIA V5 Release 19 through organized, graphically rich, step-by-step instructions on the software's basic processes and tools. This book is not intended to be a reference guide. The lessons in this workbook present basic real life design problems along with the workbenches, toolbars, and tools required to solve these problems. Each lesson is presented with step-by-step instructions. Although most of the steps are detailed for the beginner, the steps and processes are numbered and bolded so the more experienced user can go directly to the subject area of interest. Each lesson consists of an introduction, objectives, an introduction to the workbench and toolbars used in the lesson, step-by-step instructions, and concludes with a summary. Review questions and additional practice exercises are at the end of each lesson. Table of Contents 1. Introduction to CATIA V5 2. Navigating the CATIA V5 Environment 3. Sketcher Workbench 4. Part Design Workbench 5. Drafting Workbench 6. Drafting Workbench 7. Complex Parts & Multiple Sketch Parts 8. Assembly Design Workbench 9. Generative Shape Design Workbench 10. Generative Shape Design Workbench 11. DMU Navigator 12. Rendering Workbench 13. Parametric Design

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This workbook is intended to be a natural continuation of the CATIA V5 Workbook and covers a select group of advanced CATIA V5 workbenches: Sketcher, Part Design, Assembly Design, Drafting, Generative Stress Analysis, Sheet Metal Designer, Kinematics, Prismatic Machining and Knowledgeware Tools. Table of Contents Introduction to Advanced CATIA 5 Lesson 1 - Knowledgeware Lesson 2 - DMU Kinematics workbench Lesson 3 - Generative Structural Analysis workbench Lesson 4 - Generative Sheet Metal Design workbench Lesson 5 - Prismatic Machining workbench Terms and Definitions

This workbook is an introduction to the main Workbench functions CATIA V5 has to offer. The book's objective is to instruct anyone wanting to learn CATIA V5 through organized, graphically rich, step-by-step instructions on the software's basic processes and tools. This book is not intended to be a reference guide.

"[This] is a collection of tutorials meant to familiarize the reader with CATIA's mechanical design workbenches. The reader is not required to have any previous CATIA knowledge."--P. i.

The objective of this tutorial book is to expose the reader to the basic FEA capabilities in CATIA V5 Release 21. The chapters are designed to be independent of each other allowing the user to pick specific topics without the need to go through the previous chapters. However, the best strategy to learn is to sequentially cover the chapters. In this workbook, the parts created in CATIA are simple enough they can be modeled with minimal knowledge of this powerful software. The reason behind the simplicity is not to burden the reader with the CAD aspects of the package. However, it is assumed that the user is familiar with CATIA V5 Release 21 interface and basic utilities such as pan, zoom, and rotation. The tutorials are based on release 21; however, other releases can also be used with minor changes. Typically, the differences are not even noticed by a beginner.

Write powerful, custom macros for CATIA V5 CATIA V5 Macro Programming with Visual Basic Script shows you, step by step, how to create your own macros that automate repetitive tasks, accelerate design procedures, and automatically generate complex geometries. Filled with full-color screenshots and illustrations, this practical guide walks you through the entire process of writing, storing, and executing reusable macros for CATIA® V5. Sample Visual Basic Script code accompanies the book's hands-on exercises and real-world case studies demonstrate key concepts and best practices. Coverage includes: CATIA V5 macro programming basics Communication with the environment Elements of CATParts and CATProducts 2D wireframe geometry 3D wireframe geometry and surfaces Solid features Object classes VBScript commands

The objective of this tutorial book is to expose the reader to the basic FEA capabilities in CATIA V5 Release 20. The chapters are designed to be independent of each other allowing the user to pick specific topics without the need to go through the previous chapters. However, the best strategy to learn is to sequentially cover the chapters. In this workbook, the parts created in CATIA are simple enough they can be modeled with minimal knowledge of this powerful software. The reason behind the simplicity is not to burden the reader with the CAD aspects of the package. However, it is assumed that the user is familiar with CATIA V5 Release 20 interface and basic utilities such as pan, zoom, and rotation. The tutorials are based on release 20; however, other releases can also be used with minor changes. Typically, the differences are not even noticed by a beginner.

The objective of this tutorial book is to expose the reader to the basic FEA capabilities in CATIA V5. The chapters are designed to be independent of each other allowing the user to pick specific topics without the need to go through the previous chapters. However, the best strategy to learn is to sequentially cover

the chapters. In this workbook, the parts created in CATIA are simple enough that can be modeled with minimal knowledge of this powerful software. The reason behind the simplicity is not to burden the reader with the CAD aspects of package. However, it is assumed that the user is familiar with CATIA V5 interface and basic utilities such as pan, zoom, and rotation. The tutorials are based on release 15; however, other releases can also be used with minor changes. Typically, the differences are not even noticed by a beginner. The workbook was developed using CATIA in a windows XP environment. Nevertheless, it can be used for NT and UNIX platforms without any changes.

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