

Catia V5 Workbook Release V5 6r2013

Using the CATIA V5-6R2018: Introduction to Modeling learning 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 learning guide, you will have acquired the skills to confidently work with CATIA V5, and gained an understanding of the parametric design philosophy of CATIA V5. It is expected that all new users of CATIA V5 need to complete this learning guide. This guide was developed using CATIA V5-6R2018, Service Pack 1. 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, Stiffeners
- Obtaining Part Information
- Generative Drafting View Creation
- Generative Drafting Dimensioning and Annotation
- Rib and Slot Features
- Multi-sections
- Solid Features
- Feature Management Using the Hide / Show, Activate / Deactivate Functions

Parent/Child Relationships and Feature Failure Resolution Assembly Design Workbench Constraint creation, assembly management, and PDM considerations Obtaining Assembly Information (Measure, Clash, and Bill of Materials) Standard Parts from Catalogs and Save Management Working with Multi-Body Models Effective Modeling Tips and Techniques Prerequisites Access to the CATIA V5-6R2018 software. The practices and files included with this guide might not be compatible with prior versions. Experience in mechanical design and drawing production is recommended.

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 sep-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.

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Do you want to learn how to write VB script macros? There are many CAD engineers, designers, and technicians who want to write macros but simply don't have time to sit down and learn everything they need to know. Through a series of example codes and tutorials I'll explain how to use and create CATScript macros for CATIA V5. No programming experience is required! This information is not featured in the user help documentation. The purpose of this text is to show beginners how they can approach different problems and for users to rewrite code shown in the examples to suite their specific needs. I'll cover core items to help teach beginners important concepts needed to create custom VB script macros for CATIA V5.

Major Nghu, the fanatic North Vietnamese officer from book 1, is back. This time, he's got many more soldiers under his command, and uses different tactics, which he believes are guaranteed to defeat the Marines and Popular Forces of Combined Action Platoon Tango Niner. He starts by violating the Christmas truce - at a time when the Marines have American women visiting for Christmas dinner! Defeating the North Vietnamese is the toughest job Tango Niner has faced, especially once Major Nghu and his forces begin targeting the civilian population of the hamlets of Bun Hoa village. Step by step, Major Nghu believes he is achieving his ultimate goal of defeating the Marines and PFs of Tango Niner. Step by step, the Marines and PFs find ways to counter him and his forces, until they meet in the ultimate battle for control of the Song Du Ong river valley.

CATIA V5 Workbook Release 19

Introduction to Modeling

Thijo - Saga of a Norseman

Release 17

Introduction to CATIA V5 Release 19

Catia V5-6R2015 Basics

Join Thijo, a young Scandinavian farm boy, on his childhood adventures as he meets new friends and challenges throughout daily Norwegian life. Through hard days of harvest labor and deadly winter blizzards, Thijo journeys from boyhood pleasures to learning what it means to take his place among the men of the

North. Full of child-friendly adventure and excitement, Thijo - Saga of a Norseman is a book that you and your children will want to read again and again!

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.

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. This textbook explains how to perform Finite Element Analysis using the Generative Structural Analysis workbench in CATIA V5. CATIA is a three dimensional CAD/CAM/CAE software developed by Dassault Systems, France. This textbook is based on CATIA V5 Release 21. Users of earlier releases can use this book with minor modifications. It is assumed that readers of this textbook are familiar with creating parts and assemblies in CATIA V5. However, any persons not familiar with CATIA V5 modeling and assembly but interested in FEA can learn through the step by step processes laid out in this textbook, such as naming a part file, creating a 3D model for analysis or defining an FE model. Each process is accompanied by illustrations. Each chapter deals with a major topic in FEA and proceeds with an analysis procedure using CATIA V5 Structural Analysis. At the end of each chapter the author explains the meaning of the results and recommends additional topics to be considered. Engineers and mechanical engineering students are highly recommended to read this textbook to increase their knowledge of FEA by using CATIA V5 Generative Structural Analysis. Topics covered in this textbook - General concepts of FEA - Singularity in static analysis - Effects of fillets and stiffeners - Bearing loads and reflective symmetry - Rotational loads and cyclic symmetry - Use of a coordinate system in defining boundary conditions and loads - Using two dimensional and one dimensional elements - Connections: Seam weld, rigid, bolt, pressure fit and contact - Applying loads with enforced displacement - Automatic mesh

adaptation - Using the temperature effect in static analysis - Buckling and normal mode analysis"

Catia V5-6r2018

Advanced Surface Design

Catia V5-6r2017

VB Scripting for CATIA

Advanced Catia V5 Workbook

Introduction to CATIA V5, Release 16

CATIA V5-6R2017 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 V5-6R2017. This book provides elaborate and clear explanation of tools of all commonly used workbenches of CATIA V5-6R2017. 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 Generative Shape Design explains the concept of hybrid designing of models. Also, it enable the users to quickly model both simple and complex shapes using wireframe, volume and surface features. The chapter on the FreeStyle workbench will enable the users to dynamically design and manipulate surfaces. In this book, a chapter on FEA and structural analysis has been added to help users to

analyze their own designs by calculating stresses and displacements using various tools available in the Advanced Meshing Tools and Generative Structural Analysis workbenches of CATIA V5-6R2017. The book explains the concepts through real-world examples and the tutorials used in this book. After reading this book, the users will be able to create solid parts, sheet metal parts, assemblies, weldments, drawing views with bill of materials, presentation views to animate the assemblies, analyze their own designs and apply direct modeling techniques to facilitate rapid design prototyping. Also, the users will learn the editing techniques that are essential for making a successful design. Salient Features Consists of 19 chapters that are organized in a pedagogical sequence. Detailed explanation of CATIA V5-6R2017 tools. First page summarizes the topics covered in the chapter. Hundreds of illustrations and comprehensive coverage of CATIA V5-6R2017 concepts and techniques. Step-by-step instructions that guide the users through the learning process. More than 40 real-world mechanical engineering designs as tutorials and projects. Technical support by contacting techsupport@cadcim.com. Additional learning resources at

<https://allaboutcadcam.blogspot.com> Table of Contents Chapter 1: Introduction to CATIA V5-6R2017 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 the 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 Index
This workbook is an introduction to the main Work Bench 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. CATIA V5-6R2019 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 V5-6R2019. This book provides elaborative and clear explanation of the tools of all commonly used workbenches of CATIA V5-6R2019. 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 V5-6R2019. Hundreds of illustrations and a comprehensive coverage of CATIA V5-6R2019 concepts and techniques. Additional learning

**resources at 'allaboutcadcam.blogspot.com'.
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Introduction to Surface Design

Release 5 Version 13

Catia V5-6r2014 Surface Design

The Book on Internal STRESS Release

CATIA V5 Workbook

CATIA V5-6R2017 for Designers, 15th Edition

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. 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. The workbenches covered in this workbook are Sketcher, Part Design, Drafting, Assembly Design, Generative Shape Design, DMU Navigator and Rendering/Real

Time Rendering, Knowledgeware, Kinematics, and Generative Structural Analysis.

"[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.

With the combination of Coach Melvin's Dynamic Application of Internal Awareness(tm) (DAIA) Method, Dr. Totton's 100-day method to condition your body's neural pathways to establish a new habit which then becomes automatic, and with Dr. Painter's method of committed practice of Li Family Yixingong (Standing Meditation) to produce profound results at the neurological level, novices to advanced practitioners gain the ability to access your inner core, tapping into an area that can positively affect your overall well-being, prevent stress from taking hold, and give you perpetual mental-physical rejuvenation.

CATIA V5-6R2014 for Designers is a comprehensive textbook written with the intention of helping the readers effectively use all solid modeling tools and other features of CATIA V5-6R2014. This textbook provides elaborative and clear explanation of the tools of all commonly used workbenches of CATIA V5-6R2014. After reading this textbook, 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 textbook explains the concepts through real-world examples and the tutorials used in this textbook

ensure that the users can relate the knowledge gained from this textbook with the actual mechanical industry designs.

Releases 8 & 9 ; CAD/CAM Engineering Technology

***How to program CATIA V5 macros
CATIA V5-6R2018 for Designers, 16th Edition
Catia for Design and Engineering
CATIA V5 Surface Design with Applications***

Are you tired of repeating those same time-consuming CATIA processes over and over? Worn out by thousands of mouse clicks? Don't you wish there were a better way to do things? What if you could rid yourself those hundreds of headaches by teaching yourself how to program macros while impressing your bosses and coworkers in the process? VB Scripting for CATIA V5 is the most complete guide to teach you how to write macros for CATIA V5! Through a series of example codes and tutorials you'll learn how to unleash the full power and potential of CATIA V5. No programming experience is required! This text will cover the core items to help teach beginners important concepts needed to create custom CATIA macros. More importantly, you'll learn how to solve problems and what to do when you get stuck. Once you begin to see the patterns you'll be flying along on your own in no time. Visit scripting4v5.com to see what readers are saying, like: "I have recently bought your book and it amazingly helped my CATIA understanding. It does not only help you with macro programming but it helps you to understand how the software works which I find a real advantage."

This textbook explains how to create models with freeform surfaces using CATIA V5. CATIA is a three dimensional CAD/CAM/CAE software developed by Dassault Systems, France. This textbook is based on CATIA V5-6R2014. Users of earlier releases can use this book with minor modifications. We provide files for exercises via our website. All files are in CATIA V5R20 so readers can open the

files using later releases of CATIA V5. It is assumed that readers of this textbook are accustomed to the modeling tools and processes in how to construct solid models in CATIA V5. For basic modeling, assembly and drafting techniques, refer to the textbook written by the author. This textbook is suitable for anyone who are interested in learning how to create and use the freeform surface in constructing 3D models using CATIA V5. Topics covered in this textbook - Chapter 1: Introduction to Surface Design - Chapter 2: Creating a Freeform Surface in a Solid Body - Chapter 3 and 4: Creating Reference Elements and Curves - Chapter 5 through 9: Creating Freeform Surfaces with various Commands - Chapter 10: Analyzing Surface Quality - Chapter 11 through 16: Modeling Projects (Cup Holder, Router Stand, PET Bottle, Lamp Shade, Classical Handset, Bumper Surface of Audi Q5)"

Are you awed by the smell of flowers or the busy buzz of the bees? Are you awed by the all of the snow or the rustle of the trees? In *The Book of Awe*, readers are reminded to take a minute and see the beauty in the everyday things around them.

CATIA V5-6R2015 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.

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Get Powerful Health and Nutritional Secrets

A Nghu Day Dawns

Using Catia® V5

CATIA V5 Workbook, Releases 14 & 15

CATIA V5 Design Fundamentals

The Night Fighters, Book 5

This textbook explains how to create models with freeform surfaces using CATIA V5. CATIA is a three dimensional CAD/CAM/CAE software developed by Dassault Systèmes, France. This textbook is based on CATIA V5-6R2014. Users of earlier releases can use this book with minor modifications. We provide files for exercises via our website. All files are in CATIA V5R20 so readers can open the files using later releases of CATIA V5. It is assumed that readers of this textbook are accustomed to the modeling tools and processes in how to construct solid models in CATIA V5. For basic modeling, assembly and drafting techniques, refer to the textbook written by the author. This textbook is suitable for anyone who are interested in learning how to create and use the freeform surface in constructing 3D models using CATIA V5.

CATIA V5-6R2015 for Designers is a comprehensive textbook written with the intention of helping the readers effectively use all solid modeling tools and other features of CATIA V5-6R2015. This textbook provides elaborative and clear explanation of the tools of all commonly used workbenches of CATIA V5-6R2015. After reading this textbook, 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 textbook explains the concepts through real-world examples and the tutorials used in this textbook ensure that the users can relate the knowledge gained from this textbook with the actual

mechanical industry designs. In this edition, a chapter on Generative Shape Design has been added that explains mechanical engineering industry examples.

What does every mile mean to you? When you hit the trails, the road, the track or the treadmill, what does each mile mean? A group of runners and walkers from around the world share their stories as they let us know what every mile matters mean to them. Get ready to be inspired.

This textbook explains how to create solid models, assemblies and drawings using CATIA V5. CATIA is a three dimensional CAD/CAM/CAE software developed by Dassault Systèmes, France. This textbook is based on CATIA V5 Release 21. Users of earlier releases can use this book with minor modifications. We provide files for exercises via our website. All files are in Release 19 so readers can open the files using later releases of CATIA V5. It is assumed that readers of this textbook have no prior experience in using CATIA V5 for modeling 3D parts.

This textbook is suitable for anyone interested in learning 3D modeling using CATIA V5. Each chapter deals with the major functions of creating 3D features using simple examples and step by step self-paced exercises. Additional drawings of 3D parts are provided at the end of each chapter for further self exercises. The final exercises are expected to be completed by readers who have fully understood the content and completed the exercises in each chapter. Topics covered in this textbook - Chapter 1: Basic component of CATIA V5 software, options and mouse operation. - Chapter 2: Basic step by step modeling process of CATIA V5. - Chapter 3 through 6: Creating sketches and sketch based features. - Chapter 7: Usage of reference elements to create complex 3D geometry. - Chapter 8: Dress-up features such as fillet, chamfer, draft and shell. - Chapter 9: Modification of 3D parts to take advantage of parametric modeling concepts. - Chapter 10: Creating complex 3D parts by creating multiple bodies and applying boolean operations. -

Chapter 11: Copying or moving geometrical bodies. - Chapter 12 and 13: Constructing assembly structures and creating or modifying 3D parts in the context of assembly. - Chapter 14 and 15: Creating drawings for parts or assemblies. - Chapter 16: Advanced functions in creating a solid part such as a rib, stiffener and multi-sections solid.

Release 21

CATIA V5-6R2019 for Designers, 17th Edition

Releases 8 & 9

CATIA V5 FEA Tutorials

CATIA V5 Workbook Release V5-6R2013

CATIA V5

CATIA V5-6R2018 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 V5-6R2018. This book provides elaborative and clear explanation of the tools of all commonly used workbenches of CATIA V5-6R2018. 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 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.

Hundreds of illustrations and a comprehensive coverage of CATIA V5-6R2018 Concepts & Techniques. Self-Evaluation Tests and Review Questions provided at the end of each chapter to help users assess their knowledge. Additional learning resources at

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Analysis Student Projects Index

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**

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

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- Lesson 3 - Generative Structural Analysis workbench**
- Lesson 4 - Generative Sheet Metal Design workbench**
- Lesson 5 - Prismatic Machining workbench**
- Terms and Definitions**

This professional how-to guide introduces Catia

users to all of the information they need for successful feature-based design and 3D computer modelling. Comprehensive coverage includes customizing toolbars, creating assemblies models, interacting with 3D solid model features and more.

**Learn how to Write Across
CATIA V5 FEA Tutorials Release 20
Release 16**

Sketcher Workbench, Part Modeling, Assembly Design, Drafting, Sheet Metal Design, and Surface Design

**CATIA V5R20 for Designers
VB Scripting for CATIA V5**

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

The CATIA V5-6R2018: Introduction to Surface Design learning guide introduces the fundamentals of creating wireframe and surface geometry. This guide takes an in-depth look at process-based modeling techniques used to develop robust and flexible surface geometry. With the design intent as the focus, you learn about shape and continuity settings for simple and complex geometry types.

Topics Covered Surfacing terminology Surface design process Creating wireframe geometry Creating simple surfaces Creating complex surfaces Performing operations on wireframe and surface geometry Working with surface geometry in the Part Design Workbench Geometrical Element Management Surface Fillets Boundary Representations Best practices for surface modeling Prerequisites

Access to the V5-6R2018 version of the software, to ensure compatibility with this guide. Future software updates that are released by Dassault Systèmes include changes that are not reflected in this guide. The practices and files included with this guide might not be compatible with prior versions (i.e., V5-6R2017). Completion of the CATIA V5-6R2018: Introduction to Modeling course is recommended.

The CATIA V5-6R2017: Advanced Surface Design learning guide expands on the knowledge learned in the CATIA: Introduction to Surface Design learning guide by covering advanced curve and surface topics found in the Generative Shape Design Workbench. Topics include: advanced curve construction, advanced swept, blend and offset surface

construction, complex fillet creation, and the use of laws. Curve and surface analysis are introduced to validate the student's geometry. Tools and methods for rebuilding geometry are also discussed. As with the CATIA:

Introduction to Surface Design learning guide, meeting model specifications (such as continuity settings) remains forefront in introducing tools and methodologies. Topics Covered Surface Design Overview Advanced Wireframe Elements Curve Analysis and Repair Swept Surfaces Blend Surfaces Adaptive Sweep Laws Advanced Surface Fillets Alternative Filleting Methods Duplication Tools Knowledge Templates Surface Analysis and Repair Offset Surfaces Project Exercises Prerequisites CATIA V5-6R2017: Introduction to Surface Design is recommended.

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.

Catia V5-6r2014 for Designers
Macro Programming with Visual Basic Script
Advanced CATIA V5 Workbook
A Step by Step Guide
The Book of Awe
A Hands-on Tutorial Approach