Chapter 1: CATIA V5 Fundamentals

1.1] Welcome to CATIA V5
   a. CATIA V5, the New Generation
   b. The Windows Philosophy
   c. PLM (Product Life Cycle Management)
   d. PLM in Practice
   e. CATIA Within PLM
   f. What is CATIA V5?
   g. Design Intent
   h. How features affect Design Intent?

1.2] The CATIA V5 Environment
   a. CATIA User Interface
   b. The Workbench Concept
   c. Workbench Concept: Recommendations
   d. Introduction to V5 Documents

1.3] Launching CATIA and Manipulating Objects
   a. Starting CATIA V5
   b. Menus and Toolbars
   c. Manipulating Objects
   d. Manipulating Objects: Recommendations
   e. Accessing the Toolbars and Dialog Boxes
   f. What to do when a Tool cannot be found?
   g. CATIA User Interface: Recommendations
   h. Saving and Closing Documents
   i. Starting and Working with CATIA: Recap Exercises

1.4] The Specification Tree
   a. What about the Specification Tree
   b. Manipulating the Tree
   c. To Sum Up

1.5] Objects Visualization
   a. Selecting objects with the Mouse
   b. Hiding and Showing Objects
   c. The Object/Action and Action/Object Approaches
   d. Using Dialog Boxes
   e. Moving Objects with Mouse
   f. What is the Compass?
   g. Graphic Properties
   h. Changing the Graphic Properties
   i. Rendering Styles
   j. Objects Visualization: Recap Exercises
   k. CATIA User Companion
   l. Help Documentation
   m. Message Bar

1.6] Profile Creation
   a. Create a new Part
   b. Select and Appropriate Sketch Support
c. Create Sketched Geometry
d. Constrain the Sketch
e. Create Pad Feature
f. Save and Close the Document

1.7] Basic Features
a. Determine a suitable Base Feature
b. Create Pad and Pocket Features
c. Create Holes
d. Create Fillets and Chamfers

1.8] Additional Part Features
a. Create Feature Profiles and Axis System
b. Create Multi-profile Sketch Features
c. Create Wireframe Geometry
d. Create Shaft and Groove Features
e. Shell the Model

1.9] Dress-up Features
a. Apply a Draft
b. Create a Stiffener
c. Create Threads and Taps
d. Edit Features

1.10] Reusing Data
a. Duplicate Features
b. Transform a Body
c. Copy and Paste the Data
d. Insert Data from a Catalog

1.11] Finalizing Design Intent
a. Apply Material Properties
b. Analyze the Model
c. Create Formulas and Parameters

1.12] Assembly Design
a. Create a new CATProduct
b. Assemble in the Base Component
c. Manipulate the position of the Component
d. Assemble and Fully Constrain Components
e. Save the Assembly

1.13] Designing in Context
a. Open an existing assembly
b. Insert a new model
c. Create a Sketch in Context
d. Create Assembly-Level Features

1.14] Drafting:
a. The Drawing Environment
b. Drafting Toolbars and Objects
c. Start a New Drawing
d. Setting the Drawing Sheet Format and Drafting Standards
e. Starting a Drawing with a Blank Sheet
f. Sheet Properties
g. Drawing Title Blocks
h. 2D Catalogs
i. Inserting Catalog Items
j. Create Views
k. Types of Views
l. Using the Compass
m. Projection Plane / Views
n. Using Wizard
o. CGR Views
p. Dimensions and Annotations
q. Section Views and Section Cuts
r. Detail, Clipping, Broken, Breakout and Auxiliary View
s. Modifications
t. Check links to 3D Parts
u. Printing

1.15] Exercises and Workshops

Chapter 2: CATIA V5 Mechanical Design Expert

2.1] Introduction
i. Introduction
j. Review the User Interface
k. CATIA Workbenches
l. Part Design Workbench
m. Assembly Design Workbench
n. Generative Shape Design Workbench
o. Importance of Parent/Child Relationships
p. Investing the Model
q. What is Define in Work Object?
r. Defining in Work Objects
s. Organizing a Solid Model
t. Model Organization
u. Geometrical Sets
v. Ordered Geometrical Sets
w. Hybrid Design

2.2] Design Complex Parts
a. Design Intent
b. Create Advanced Sketch-Based Features
c. Multi-Sections Solid
d. Create Advanced Drafts
e. Advance Dress-Up Features
f. Use the Multi-Body Method
g. Create Multi-Model Links

2.3] Surface Design
a. Access the Surface Design Workbench
b. Create the Reference Geometry
c. Create the Basic Surface Geometry
d. Create the Complex Surface Geometry
e. Perform Operations on Surfaces
f. Solidify the Model

2.4] Analyze and Annotate Parts
a. Analyze the Part
b. Create 3D Constraints
c. Annotate the Part

2.5] Sharing Information
a. Create a Power Copy
b. Create Parameters and Formulas
c. Create a Design Table
d. Create a Catalog

2.6] Assembly Design
a. Manage the product structure
b. Create flexible sub-assemblies
c. Analyze the assembly
d. Create scenes
e. Create annotations
f. Generate reports

2.7] Contextual Design
a. Clarify the display
b. Create Contextual Parts
c. Create Assembly-Level Features
d. Manipulate the Contextual Components
e. Save the Contextual Models

2.8] Complex Assembly Design
a. Create a Skeleton Model
b. Create the Published Elements
c. Use the Published Elements

2.9] Exercises and Workshops

Chapter 3: CATIA Part Design Added Exercises
a. Connector Housing
b. Bike Rear Lever
c. Fitting
d. Bottom Case
e. Angle Bracket
f. Meat Mincer Screw
g. Hinge
h. Car Jack Support
i. Piston
j. Side Toolhead
k. Tee Fitting

Chapter 4: CATIA Product Design Added Exercises
a. Connector Assembly Exercise
b. Vice Assembly Exercise
c. Flexible Assemblies Exercise
d. PC Speaker Assembly Exercise

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