



Catia V5

Organizations that utilize the Catia V5 Computer Aided Design (CAD) system to develop their products are using a cutting edge system that help analyze, validate and document products utilized throughout the complete product life cycle. This course is focused on training the existing Catia V5 user by expanding their skills and techniques to further utilize advanced features of the CAD system.

This course consists of multiple modules that are structured in a pedagogical sequence, covering the Part, Assembly, Drafting, and Generative Sheet metal Design workbenches of CATIA V5.

Participants will bring information on specific company projects to be worked on during this training for real application of these concepts, tools and techniques.

- First, every module is covered and they begin with a section that provides a detailed explanation of the commands and tools in Catia V5R19.
- Next, the command section is followed by tutorials that are created using these commands. This approach allows the student to use the text initially as a learning tool and then later as reference material.
- Lastly, the students will work on specific projects that show the preferred method of application of Catia V5R19 for their job requirements.



Course Syllabus

I. IDENTIFYING INFORMATION

Course: Catia V5

Prerequisite: Design or Engineering experience
Basic computer skills

Time Frame: 40 total contact hours, 10 modules will be covered

Instructor: Lee Kittredge
Dassault Certified Catia V5 instructor
20 years in the CAD field

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II. REFERENCE MATERIALS

1. CATIA V5 for Designers by: Sham Tickoo and Vivek Singh

III. COURSE GOALS AND OBJECTIVES

1. Recognize and navigate toolbars
2. Access workbenches
3. Efficiently create fully constrained sketches
4. Create and modify solid parts
5. Create basic surfaces



IV. METHODOLOGY

This course provides the solid fundamentals of the CAD tool to prepare the student for more specific and advanced functions. Each module will introduce new material that will prepare the student for the projects to be completed.

Lectures

Each detailed subject will be presented in a lecture format outlining the theory and standardized accepted methodology. A PDF file of the lecture material will be provided for the student's personal use as reference material. Lecture note outlines will be distributed to the students for each lecture to help the student capture personal notes.

Specific Industry Examples

Real life industry examples will be covered that detail out the application of the theory to demonstrate how different companies apply these tools and techniques. This will give the students a clear understanding of how and why these techniques are utilized at different companies and industries in different manners.

In-Class Assignments

Using the theory and industry examples the student will conduct several projects that outline each key principal on in-class projects. These projects will increase in complexity as the students further develop their skills in applying these tools and techniques.



V. **COURSE OUTLINE AND ASSIGNMENTS**

Module 1: Introduction to Catia V5

Workbenches	Discussion
Toolbars	Discussion
Specification Tree	Discussion
Compass	Discussion
Mouse Usage	Discussion
File Operations	Discussion

Module 2: Drawing Sketches in the Sketcher Workbench-I

Invoking the Sketcher Workbench	Discussion
Units, Grid Settings	Discussion
Construction and Standard elements	Discussion
Drawing Sketch objects I	Discussion
Inferred Constraints	Discussion
Exiting the Sketcher Workbench	Discussion
View Manipulation	Discussion
Chapter 2 Tutorials 1-3	Assignment

Module 3: Drawing Sketches in the Sketcher Workbench-II

Drawing Sketch Objects II	Discussion
Modifying sketch Objects	Discussion
Sketch Transformations	Discussion
Chapter 3 Tutorials 1-3, Exercise 1	Assignment

Module 4: Constraining Sketches and Creating Base Features

Constraining Sketches	Discussion
Geometric Constraints	Discussion
Dimensional Constraints	Discussion
Sketch Analysis	Discussion
Creating an Extruded Solid Using the Pad Feature	Discussion
Creating a Revolved Solid Using the Shaft Feature	Discussion
Assigning a Material to the Model	Discussion
Chapter 4 Tutorials 1 and 2, Exercises 1 and 2	Assignment



Module 5: Reference Elements and Sketch-Based Features

Reference Elements	Discussion
Drafted Filleted Pad Features	Discussion
Multi-Pad Features	Discussion
Pocket Features	Discussion
Drafted Filleted Pocket	Discussion
Multi-Pocket Features	Discussion
Groove	Discussion
Extruding and Revolving Faces	Discussion
Projecting 3D Elements	Discussion
Chapter 5 Tutorial 1-3	Assignment

Module 6: Creating Dress-Up and Hole Features

Hole features	Discussion
Chamfer	Discussion
Fillets	Discussion
Draft	Discussion
Shell	Discussion
Chapter 6 tutorials 1 and 2, Exercise 1	Assignment

Module 7: Editing Features

Editing Features	Discussion
Cut, Copy and Paste	Discussion
Deleting Features	Discussion
Deactivating Features	Discussion
Defining the Work Object	Discussion
Reordering Features	Discussion
Parent Child Relationships	Discussion
Update Diagnostics	Discussion
Measuring Elements	Discussion
Chapter 7 Tutorials 1 and 2, Exercise 1	Assignment

Module 8: Transformation Features and Advanced Modeling Tools-I

Translating and Rotating Bodies	Discussion
Symmetry and Mirror Features	Discussion
Patterns	Discussion
Scale	Discussion
Parts with Multiple Bodies	Discussion
Boolean Operations	Discussion
Stiffener Features	Discussion
Chapter 8 Tutorials 1 and 3, Exercise 1	Assignment



Module 9: Advanced Modeling Tools-II

Rib	Discussion
Slot	Discussion
Multi-section Solids	Discussion
Power copy	Discussion
Chapter 9 Tutorials 1 and 3, Exercise 1	Assignment

Module 10: Wireframe and Surface Design

Surfacing Workbenches	Discussion
Wireframe geometry	Discussion
Extruded Surfaces	Discussion
Revolved Surfaces	Discussion
Offset Surfaces	Discussion
Swept Surfaces	Discussion
Fill Surface	Discussion
Multi-section Surfaces	Discussion
Blended Surfaces	Discussion
Split, Trim and Join	Discussion
Chapter 10 Tutorials 1 and 2, Exercise 1	Assignment