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CATIA Automation

Languages: VB.NET, C#, C++, MFC

1. The Component Application Architecture: CAA

Learn to declare the CATIA object variables and the mechanism of accessing CATIA programmatically. Traversing the CAA object model and manipulating basic properties and interface elements.

Start Show Stop CATIA: Learn to control CATIA at will. Learn to invoke, hide, show CATIA, read-write various properties of CATIA objects and quit CATIA.

2. Detecting the Workbench

Learn to detect which document type is currently open using the CAA and .Net techniques.

3. CATIA Documents: Open, Close, Create, Save & Export CATIA Docs

Learn to create new files, open existing files, save a part file, save as and export CATIA documents to other formats. How to close selective and all files programmatically.

4. File Handling using WinAPI & CATIA API

Learn to invoke CATIA's standard file Open and Save dialogs with preview. Learn to show Windows standard file open/save dialogs to select CATIA files.

5. Managing Sheets In Drawings

Learn about handling sheets in a drawing, counting sheets, sheet info, switching between sheets, accessing sheets, deleting and exporting sheets to DXF etc.

6. Drawing Views

Learn to create standard views on drawing sheets and critical aspects like spacing between views, etc.

7. Processing Imported Legacy Data

Learn to handle Dimensions In CATDrawings: Many times migrated or imported geometry needs to be tweaked. Learn how to setup console programs that run unattended, picking up the right entities in a CATDrawing and manipulate them, saving back the files properly.

8. Handling Text In CATDrawings

Title Block Updater: Learn to handle and access text objects in a drawing document and how to update text in a title block to maintain consistent font and spacing across drawings. Learn how to meticulously design user-friendly interfaces for CATIA automation apps.

9. CATIA Part Features

Traversing Features of a CATIA Model: learn to traverse through all features of a part and access the feature properties.

10. Advanced Part Features and Selection Sets

Learn to create a Data Archiver for CATIA: Learn to build a real-world program for archiving CATIA files.

11. Parametric Parts

Designing a Shaft with a Keyway: Learn to parametrically control a CATPart. This is perhaps the most widely used CATIA technique in design automation.

12. Morphing Parts - Part Families

Creating a Super Shaft: Learn to intelligently build required features of a family of parts into a model and how to manipulate them to get the desired configuration of a part.

13. CATIA Products and Physical Properties

Material Estimation for Spray Painting: Learn to traverse through all components of a CATIA Product and manipulate them. Also learn how to extract physical properties like area, mass, volume, etc.

14. CATIA Product Generation

Inserting Parts Into a Product: Learn how to build a Product from scratch by assembling CATIA part files into an empty assembly, learn to setup pointers to the inserted components, manipulate them and move them around in the product, scale and rotate parts in an assembly.

15. Product Configurator

Creating a CATIA Product Configurator - Product configurators are multifunctional, computer-aided systems that function as an interface between sales and product design. These automatically generate product documents, for example, bills of material, production drawings, product cost, virtual 3D models, quotations, routings, product images, reports & catalogs, sales & approval drawings and so on. Learn to setup part libraries, present users with a choice of parts from the library to pick and traverse through a CATIA product document and build the product configuration as specified by user. This is a lite version of a paid project.

16. Batch Processing

Learn to setup a batch processor for handling CATIA documents. Learn to use the Windows standard 'Browse for Folder' dialog and gather all files of a specified type into a .Net collection and how to selectively remove undesired files from a listbox. Learn to traverse through all views in all the CATDrawings in a specified folder and update them. This can be useful before batch exporting CATDrawings to other file formats like dwg, dxf, etc.

Other eBooks and Training Programs from CADVertex:

1. **CATIA Automation:** VBA, VB 6, VB.Net CSharp, C++, MFC
2. **SolidWorks API:** VBA, VB 6, VB.Net CSharp, C++, MFC
3. **Solid Edge API:** VB 6, VB.Net CSharp, C++, MFC
4. **NX Open:** VB.Net CSharp
5. **Knowledge Fusion**
6. **Inventor API:** VB 6, VB.Net CSharp, C++, MFC
7. **AutoCAD API:** VB 6, VB.Net CSharp, Visual LISP, DCL
8. **Open CASCADE:** Programming using C++ and MFC
9. **cMayoCAD:** Learn to build a new CAD program using a Geometric Modeling Kernel.

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