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NX Open

Languages: **VB.Net or C#**

14 topics with labs and projects

1. Accessing NX Programmatically

Learn to declare the NX object variables and the mechanism of accessing NX. Loading and Unloading a library. Traversing the NX object model and manipulating basic properties and interface elements.

2. NX User Interface Elements

Learn to use various commonly used NX user interface elements which are also available in Win Forms.

3. NX File Handling - Open, Close, Create, Save, & Export NX Files

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

4. Drawing in 2D

Learn to create basic 2D entities like points, lines, arcs, etc. Learn to create annotations, set positioning and various properties. Learn to gather static input data from user for positioning.

5. Managing NX Sheets

Learn about handling sheets in a drawing, counting sheets, sheet sizes, switching between sheets, setting/accessing work sheet, sheet names, checking out-of-date sheets. Learn key concepts of .Net collections and NX Open collections.

6. Drawing Interactively in 2D

Learn to create 2D entities and positioning by gathering dynamic input data from user. Learn about the NX selection manager.

7. NX Drawing Views

Learn to create standard views on drawing sheets. Learn about the NX UI class. Understanding and using the Selection Manager. Selecting points on screen, etc.

8. Extracting Geometry Data

Learn how to traverse through NX native and imported geometry and extract geometric information to calculate cost of Glow sign shapes. Learn about handling layer.

9. Handling Text In NX Drawings

Title Block Updation – 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 dialogs for NX automation apps.

10. NX Part Features

Learn to traverse through all features of a part and access the feature properties.
Learn to create 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.

11. NX Parametric Parts

Designing Parametric Parts – Learn to parametrically control a Part. Master the most widely used NX technique in design automation.

12. Traversing NX Assemblies

Learn to traverse the assembly structure at the top level and access required data of the components of an assembly.

13. NX Part Attributes

Building an in-house Product Coster – Learn about the NX attributes dialog and how to use it in-depth. Also learn how to setup and access the custom properties in NX documents to create a product coster.

14. Batch Processing Techniques

A Batch DXF/STEP Exporter – export all NX files in a specified folder to dxf/step format. Learn to use the Windows standard 'Browse for Folder' dialog, learn to gather all files of specified extension into a .Net collection and how to selectively remove undesired files from a listbox.

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 or CSharp
5. **Knowledge Fusion**
6. **UFunc**
7. **Inventor API:** VB 6, VB.Net CSharp, C++, MFC
8. **AutoCAD API:** VB 6, VB.Net CSharp, Visual LISP, DCL
9. **Open CASCADE:** Programming using C++ and MFC
10. **cMayoCAD:** Learn to build a new CAD program using a Geometric Modeling Kernel.

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