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Nick Weisenberger



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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. Includes seven step by step how to tutorials.

Catia V5-6r2017 ASCENT - Center for Technical Knowledge, 2020-01-06. The CATIA V5 6R2017 Visual Basic Automation learning guide provides you a good understanding of the different ways to automate tasks using CATIA macros and Visual Basic programming. Using hands on practices you will use VB programming to work with parts, assemblies, drawings, selections, parameters and formulas, graphic properties and to exchange data with Microsoft Excel. This guide was written against CATIA V5 6R2017 Service Pack 1. Topics Covered: CATIA V5 Object Model, Creating Part Design and Shape Design features, Working with Product Structure and Assembly Design, Scripting Drawing Views, Frames and Title Blocks, Deleting, Cutting, Copying, Pasting CATIA objects, Interactive Selections, Communication with MS Office. Prerequisites: Visual Basic programming and working knowledge of CATIA.

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