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Use the following steps to open an existing document within CATIA: Select File > Open, or click the Open icon. Completely re-written since CATIA v4 and still under development! CATIA v5 is a native Windows application! Lecture TIA VCATIA is a three dimensional CAD/CAM/CAE software developed by Dassault Systèmes, France. Additionally, delve into the Dassault Systèmes CAA Encyclopedia for developer's guides, covering V5 & V6 development toolkits Insights. Science. Based on Variational/ Parametric technology! Explore our Dassault Systèmes user assistance guides and learn more on V6, 3DEXPERIENCE platform applications and SIMULIA Established products CATIA V5 Workbook Figure Figure Figure Figure The Current Active CATIA V5 Document This area of the screen displays the name of the current active CATIA V5 document. For a close up view with document circled CATIA V5 Workbook Objectives The objective of this workbook is to instruct anyone who wants to learn CATIA V5 through organized, graphically rich, step-by-step instructions on the basic processes and tools provided by CATIA VThe objectives of this lesson are listed belowProvide suggestions on how to best utilize this workbook CATIA Overview. This textbook is based on CATIA V5 Release Users of earlier Introduction to CATIA VCATIA V5 Platforms CATIA V5 is configured into with three different platforms. In the Files of type list, select the type of document you want to open The platforms P1, P2 and P3 designate the design level available to CATIA V5 Fundamental: – Family of dedicated applications – Natively integrated into V5 architecture – Easy to use and to deploy Unique Powerful Enabler: – Automate CATIA V5 reduces the learning curve, as it allows the flexibility of using feature-based and parametric designs. Design. CATIA V5 provides three basic platforms: P1, P2, and PP1 is Explore our Dassault Systèmes user assistance guides and learn more on V6, 3DEXPERIENCE platform applications and SIMULIA Established productsV Explore Dassault Systèmes' online user assistance collections, covering all V6 and 3DEXPERIENCE platform applications, as well as SIMULIA established productsCATIA V5RFundamentals User Interface Below is the layout of the elements of the standard CATIA application. Cloud. A, Menu Commands B. Specification Tree C. Filename and extension of current document D. Icon of the active workbench E. Toolbars specific to the active workbench F. Standard Toolbar G. Compass H. Geometry area A C E F B H G D General Explore Dassault Systèmes' online user assistance collections, covering all V6 and 3DEXPERIENCE platform applications, as well as SIMULIA established products. Browse to the file's location using the Look in list of the File Selection dialog box. Step explains how to create your favorite work benchesSelect the Start tool in the top left side of the screen, reference Figure This will bring up a pull down menu that includes all of the CATIA V5 Functions such as Infrastructure, Mechanical Design, Shape etc., and their work benches Opening an Existing Document. Manufacturing. This course CATIA V5 Dr Ahmed Kovacevic City University London School of Engineering and Mathematical Sciences Room CM, Phone: E-MailOnline Help. CATIA v5 is an Integrated Computer Aided Engineering tool: Incorporates CAD, CAM, CAE, and other applications! Virtual Twin Experience. The active CATIA V5 document shown in Figure is the default name (t) for a CATPart document. User friendly icon based graphical user interface! CATIA V5 Fundamentals course is to teach you how to build parts and assemblies in CATIA, and how to make simple drawings of these parts and assemblies. Sustainability.