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The reader will learn how to sketch and constrain very simple to very complex 2D profiles. CATIA v5 is an Integrated Computer Aided Engineering tool: Incorporates CAD, CAM, CAE, and other applications! Software Overview. User friendly icon based graphical user interface! CATIA v5 is a native Windows application! 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 CATIA VDrafting FundamentalsFree download as PDF File.pdf), Text File.txt) or view presentation slides online We are the 3DEXPERIENCE company Dassault Systèmes It shows how to automatically generate associative drafting from 3d mechanical parts produced with CATIA Version 5Isometric View. Part Design and Sketching. CATIA V5RFundamentals User Interface Below is the layout of the elements of the standard CATIA application. The user task section of the book describes in detail the procedures for using all of the features of the Structure Design product. The user task section of the book describes in detail the procedures for using all of the CATIA Drafting CATIA® V5RDrafting Currently, it is necessary for designers to create paper drawings and layouts of their parts and assemblies. Completely re-written since CATIA v4 and still under development! Based on Variational/ Parametric technology! Getting the Most Out of this Guide To get the most out of this guide, start with the tutorial in the Getting Started section. To create an isometric view you orient your part in Part Design the way you want it and then using the appropriate icon you select the body and the isometric view is generated. LectureIntroduction. The procedures are divided into sections CATIA Overview. This course will cover Introduction. Select the isometric view icon Generative Drafting User's Guide. LectureWireframe To get the most out of this guide, start with the tutorial in the Getting Started section. Getting the Most Out of this Guide To get the most out of this Introduction. CATIA is a robust application that enables you to create rich and complex designs. LectureProduct Structure and Assembly Modelling. Change the configuration of your windows to Tile Horizontally. Tutorials Contained in This Tutorial is an introduction to Generative Drafting. Chapterfocuses on CATIA's Sketcher workbench. The aim of the CATIA V5 Fundamentals course is to teach you how to build CATIA VDrafting FundamentalsFree download as PDF File.pdf), Text File.txt) or view presentation slides online Generative Drafting User's Guide: explains how to generate drawings from 3D parts and assembly definitions. More advance Part Design.