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Get Basic Tasks Started Advanced Tasks Workbench Description Customing Glossary Index Dassault Systmes 1994-2001. All rights reserved. Prefacente MapaceAbout this ProductMore InformationWhat's New? Ottenere StartedEntering the Part Design WorkbenchCreare a PadDrafting a Face fileting an EdgeEditing the Padchewing the PartSkeeting a Circle from a Face Creating a PocketShelling the PartBasic TasksApertion a new CATPart DocumentSketch-Based Featunts-Sub-Elements of a SketchUp to Next PadsUp to Last PadsUp to Last PadUp to To Padup to Surface PadNot Normal to Sketch PadDrafted Filled PadketDrafted Filled PocketShaftGroouveHoleLocaling HolesThreat HoleRibSlotStiffenerLoftRemote Lofted Lofted Material-Up Features Lofted FilleColor Filleble Radius Filleble Draft di FilleFace FilleFace-FilleTritell Basic Draft con Parting Element Basic dalle caratteristiche riflettenti LinesVariabili Angolo DraftAdvanced DraftShellTykness ThreaadSurface-Based Features SplitThick SurfaceClose SurfaceSew Surface Transformation Features TranslationRotationSymmetry Rectangular PatternCircular PatternUser PatternExplosing Model SchernScalingReference Elements Modifica Part, Bodies and Features Reording Features ParentScanning the Part and Defining local Objects Updating Party Delating Features Costructs 3d Costructs Seting Costructs Definil in Dialog BoxModifying Construcing Costrutting Mean Dimensions sReplacing a Surfacente a BodyReplacing Name PropertiesFeatures Propertities Annotations Testual Annotations Advanced TasksAssociating BodesInsiting a BodyAssembly Intersecting Bodies Additing Bodies Additing Bodies Rimozing BodesTriming Bodies Elimina LumpToolissDraft Analytiure Curvature Analysis-Three Analysis Multi-DocumentPowerCopyCreare PowerCopiesing Salvating PowerCopies in un PowerCopies Catalogo la tua DesignCut, Copia, PastePart D. WorkbenchPart Design Menu BarSketch-Based Features Dress-Up Features Surface-Party Features Transformation Features Reference Elements Boolean OperationSkeCostructs Analysis AnnostatationsCustomingGlossaryIndexPreface The CATIA Version 5 Part Design application consente di progettare precise parti 3Dmeccanici con unàinterfaccia utente intuitiva e flessibile, dal disegno in un assemblaggio al design dettagliato iterativo. L'applicazione CATIA Version 5 Part Design permetterÀ di soddisfare i requisiti di progettazione per parti di varie complessitÀ , da un semplice rostrato. Questo The application, which combines the power of functionality-based design with the flexibility of the boolean approach, offers a highly productive and intuitive design environment with multiple design methodologies, such as post-design parametrization and local 3D. As scalable product, CATIA Version 5 Part Design can be used in collaboration with other competing or future products such as CATIA Version 5 Assembly Design and CATIAVersion 5 Generative Drafting. The largest portfolio of applications in the field is also accessible through the interoperability with CATIA Solutions Version 4 to support the complete product development process from the initial concept to the product in operation. The Part Design User Guide has been designed to show how to create a part. There are several ways to create a part and this book aims to illustrate the different phases of creation that can be encountered. Information about this product More information This product This book is intended for the user who needs to become rapidly familiar with the product CATIA-PartDesign Version 5 Release 7. The user should be familiar with CATIA basic version 5concepts such as document windows, standards and display tool bars. To get the most out of this guide, we recommend you start reading and perform the Step-Step Start. This tutorial will show you how to create a base part from scratch. The following sections deal with the processing of CATPart data, therefore the creation and modification of various types of features that it is necessary to build parts. This guide also features other PartDesign features that allow you to design complex parts. You may also want to take a look at the sections that describe the design workbench part at the end of the guide. Where to find further information on reading this book, yes to read the CATIA-Infrastructure User Guide Version 5 and CATIA-Dynamic Sketcher User Guide Version 5. The CATIA-Assembly Design Guide User Version 5 CATIA Wireframe and Surface User'sGuide User'sGuide 5 And the GUIDATIVE GUIDANCE OF THE USED Version 5 may also be useful. It also sets out the conventions used in this guide. What's new? InstrumentNew: a command for the display and analysis of threads and tapsNew: a command for the publication of publishing elements: the parent command for children allows you to change the EnHanced functions: Draft AnalysenenHanced: Curvature AnalysisNew: A command to move sketches from a body to Anothetchetned Functionality based on ANROTHETCHECE Now it can be obtained from the subelements of a sketchenhenced: Pad (Sketer) Pad (Retromarcia) Enhanced: Pocket (Different Profiles) Pocket (Sketcher) Pocket (Reverse) Enhanced: Tree (different profiles) Tree (Sketch) (Opposite side) Improved: Groove (different profiles) Groove (Sketer) Groove (back side) Improved: rib (different profiles) RIB (Sketcher) Enhanced: Slots (different profiles) Slot (Sketcher) Dress-up Features: ThreenHanced: Edge Filetto (with limiting planes, faces or surfaces), edge thread (hold edge) EDGEFILLET (setting tapes) improved: Variable radius thread (with limiting planes, faces or surfaces), variable radiusfallet (hold the edge) Improved variable radius thread (cutting tapes): front thread (with crop planes, faces or surfaces) improved: Tritannic thread (with cutting planes, faces or surfaces) improved: you can use the cone or square mode for the drawing of partsenHanced: Preview of redattiNew faces: a command for the writing of the faces using angioplamati values of different analysis: Advanced PRENCANCE: Rectangular, circular or user models allow you to duplicate the geometry of currentsSolidsboolean OperationsenHanced: you can now use bodyscustomizingNow series: customization of the local gadget before entering the detailed instructions For use of the design of the Catia part5, the tracking tutorial aims to give you a feeling of what you can do with the product. Provides astep-by-step scenario showing how to use key features. The main main activities In this section are:Inserting the Part Design WorkbenchCreating a PadCreating a FaceMaking an EdgeModifying the PadMirroring the PartCreating a Circle from a FaceCreatig a PocketCreating the PartTogether, the tasks should take about ten minutes to complete.The final part will be like this:Now, let's get to the profile sketch!En Preparing the Workbench for Parts DesignThis first task shows you how to access the Workbench for parts design.1. Select File -> New Commands (or click the New icon).The New dialog appears, allowing you to choose the type of document you want.2. Select Part in the Type List field and click on OK.The Part Design workbench is loaded and a blank CATPart document is opened.The commands for creating and modifying the functionality are available in the workbenchtoolbar. Now, let's do the following task Create a Pad.Creating a PadThis task will show you how to create a pad, i.e. extrude a profile drawn in the workbenchSketcher. For more information on this workbench, please refer to the CATIA-DynamicSketcher Version 5 User Guide.Open the GettingStarted1.CATPart document to open the required profile.The profile was created on the xy plane. It looks like this:1. Select the profile if it is not already selected and click the Pad icon.The Definition dialog box Pad will appear. Default options allow you to create a basicpad.2. If you prefer to create a larger buffer, enter 60 mm in the Length field.The application displays a preview of the buffer you want to create.3. Click OK.The pad is created. Extrusion is done in a direction normal to the drawing plane. CATIA displays this creation in the specification tree:CATIA allows you to control the display of some of the components. To learn more about components that you can display or hide, do to customize the views to tree and geometry. For more information about the pads, refer to Pad, Pad, Pad'Fino a Avanti', Pad 'Fino a Ultimo', Pad 'Fino a Piano', Pad 'Fino a Surface', Pad non Normali a Disegno Piano. show you how to drawdo. 1. Click on the project angle. The Project Definition dialog box will appear. The application displays the default direction of shooting on the part. 2. Select the face indicated by the arrow as the face to be drawn. CATIA detects all faces to draw. The selected face is now dark red, while the other faces are light red. 3. Click the Neutral Element Frame Selection field and select the upper face.

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