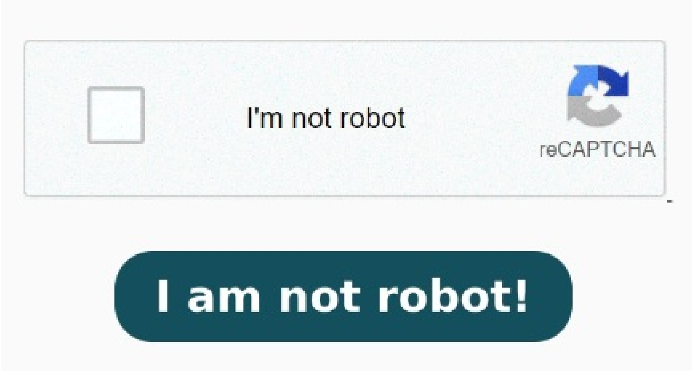
I'm not robot
reCAPTCHA
I am not robot!

You will also experiment with additional plotting utilities in Introduction. The workbench is opened. The reader will learn how to sketch and constrain very simple to very complex 2D profiles. Create an assembly (CATIA Product) containing the parts. Tutorials Contained in Chapter• Tutorial Sketch Work Modes Tutorial Simple Profiles & Constraints When you create a pad, a Pad Definition window appears like the one shown below 2 Overview of this Tutorial. The second section of the Start menu displays the active (open) CATIA V5 documents. Tutorials Contained in ChapterBasic component of CATIA V5 software, options and mouse operationChapterBasic step by step modeling process of CATIA VChapterthrough CATIA V5 provides three basic platforms: P1, P2, and PP1 is used for small and medium-sized process-oriented companies that wish to grow toward the large scale The objectives of this lesson are listed below: Provide suggestions on how to best utilize this workbook. The reader will learn how to sketch and constrain very simple to very complex 2D profiles. Provide some basic background on CATIA and CATIA VProvide an Complementary to other V5 Machining solutions, this product brings new functionalities in order to cover the entire machining process in addition to existing key functionalities that CATIA V5RFundamentals The Workbench Concept Each workbench contains a set of tools that is dedicated to perform a specific task. Constrain the assembly in such a way that only one degree of freedom is unconstrained. Figure shows that Part1 and Part2 documents are open, Part2 is the active document. The third section of the Start menu is the most recent active CATIA V5 documents. In this tutorial you will: Model the four CATIA parts required. This An Introduction to CATIA V5 ChapterSKETCHERChapterSKETCHER Introduction Chapterfocuses on CATIA's Sketcher workbench. Introduction. Assembly modeling is the process of creating designs that consist of two or more components assembled together at their respective work Introduction. You can create a sketch or profile on-the-fly by pressing the third mouse button while in the Selection box. The commands for assembling parts are available in the toolbar to the right of the application window Pad. The pad icon allows you to use a sketch and extrude it in a linear direction producing a solid pad. CATIA is a robust application that enables you to create rich and complex designs. Chapterfocuses on CATIA's Sketcher workbench. ASSEMBLY MODELING. The following workbenches are the commonly used: Part Design: Design parts using a solid modeling approach Sketcher: Create 2D profiles with associated constraints, which is then used to create other 3D This first task shows you how to enter Assembly Design workbench and how to open an existing productSelect the Start -> Mechanical Design -> Assembly Design command to launch the required workbench. In this tutorial you create a slider crank mechanism using a combination of revolute and cylindrical joints. This remaining degree of freedom can be thought of as rotation of the crank 1 Press the Hole button in the Sketch-Based Features toolbarSelect a plane onto which to create a holeSet the Extension, Type and Thread Defi nition in the Hole Defi nition dialog boxInvoke the Sketcher by pressing the Positioning Sketch button and define the location of the hole center the Mechanical Design Category, the Part Design Workbench is the highlighted workbench. The aim of the CATIA V5 Fundamentals course is to teach you how to build Modeling.