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CATIA V5R16 surface modeling - Mouse CATIA Surface-modeling Tutorial 2A - Import 2D outline drawing into Catia - Build 3D curves based on the imported drawing - Build the upper surfaces of the mouse (by Generative Shape Design) Tutorial 2B - Do the draft analysis to search any undercut portion on the upper surfaces

CATIA V5 Surface-modeling

INTRODUCTION TO CATIA V5. CATIA (an acronym of computer aided three-dimensional interactive application) developed by the French company Dassault Systèmes. CATIA INTERFACE(GUI) SPECIFICATION TREE. MOUSE CONTROLS. VIEW TOOLBAR. OTHER COMMONLY USED TOOLS: Step 2: Sketcher Module

CATIA V5 Basic Tutorial | GrabCAD Tutorials

This tutorial includes an introduction of the main features in the 3D design software package Catia V5. Beside the basic tools of 3D design, a number of exercises and examples point to different construction strategies in several applications. In addition to the primary functions, methods for the generation of solid components and assemblies are explained and executed by use of different examples.

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CATIA V5 Student Edition | 3DEXPERIENCE Edu

1. Model the four CATIA parts required. 2. Create an assembly (CATIA Product) containing the parts. 3. Constrain the assembly in such a way that only one degree of freedom is unconstrained. This remaining degree of freedom can be thought of as rotation of the crank. 4. Enter the Digital Mockup workbench and convert the assembly constraints into

CATIA V5 Tutorials - SDC Publications

Creating a Manual Explode Scene in CATIA V5. By Trisha Freeman. In the Assembly workbench, locate the "Scenes" toolbar. (If you cannot find the toolbar, go to View→Toolbar from the windows menus at the top of the screen and turn it on). When creating multiple scenes, it is always a good idea to start off by creating a default generic scene that displays the original product file with all components in their correct location.

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