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In addition to that, there are tutorials, exercises, and self-test questionnaires at the end of each chapter. These ensure that the user gains practical knowledge of each chapter before moving on to more advanced chapters. Table of Contents. 1. Getting Started with CATIA V5-6R2015. 2. Sketcher Workbench. 3. Basic Sketch Based Features. 4.

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You can access the Sketcher workbench in many ways. Two simple ways are by using the top pull down menu (Start - Mechanical Design - Sketcher), or by selecting the Sketcher icon. When you enter the sketcher, CATIA requires that you choose a plane to sketch on. You can choose this plane either before or after you select the Sketcher icon.

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CATIA V5R16 Fundamentals Create a Sketch 1. Select a planer support (e.g. datum plane, planer solid face) from the specification tree or by clicking the support directly. 2. Select the Sketcher Icon from any workbench where is possible to create a sketcher (e.g. Part Design workbench). 3. CATIA switches the current workbench to the sketcher

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This creates a "Simple" sketch (sketch, for which we do not specify the origin and orientation of the absolute axis.). To edit a sketch: Double-click the sketch or an element of the sketch geometry. You can select it in the geometry area or in the specification tree. 3D, right-click the sketch in the specification tree, Move to [sketch name] object in the contextual menu, select Edit .

### CATIA V5: Sketcher Basics 1

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