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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. The Sketcher workbench is a set of tools that helps you create and constrain 2D geometries.

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1. Start CATIA V5-6R2015. 2. Open a new part file. 3. Draw the sketch on the YZ plane, as shown in figure. 4. Click Exit workbench to complete the sketch. 5. On the Reference Elements toolbar, click the Plane icon. 6. On the Plane Definition dialog, click Plane type > Normal to curve. 7. Click on the endpoint of the sketch to define the plane location. 8.

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