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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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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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## **Sketcher Outputs in CATIA V5 - resources.rand3d.com**

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