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December 18, 2020. Posted By: Peta-Ann

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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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Written by Dickson Sham A- 1 Surface-
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assemblies and drawings. Understand how to manage parts in the context of an assembly. This is a hands-on course where students produce simple parts drawings and assemblies.

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Catia V5 : Fundamental 3D Modeling
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Upon completion of this course you will be able to: - Understand and use the CATIA V5 interface. - Plan the construction of a part in order to convey its. visual and functional aspects. - Create simple parts in CATIA V5. - Construct an

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assembly using the parts. - Produce simple drawings and assembly layouts.

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The CATIA V5 Knowledge Fundamentals is an instructor led training session, covering various topics to embed

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knowledge in CATIA V5 designs. The course is delivered by Design Rule's in-house trainers and is hosted at our Silverstone office or customer sites.

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