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Catia V6 Manual

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CATIA V6 Tutorials contains most used tips and tricks, workflows and tutorials about how to use CATIA V6 modules (Part Design, Generative Shape Design) and more.

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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

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CATIA V5 Surface-modeling

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