

## Catia V6 Manual

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### Catia V6 Manual

A simple but power-full tool is CATIA V5 is the Mass section, from where you can find very fast the main dimensions and weights of a part or of an assembly. To be more precise is very important to have assigned to each PartBody an material. You need to have on your interface active the [...]

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### CATIA V6 Tutorials - CATIA V5-V6 Tutorials

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CATIA Composites 3DEXPERIENCE® R2019x Review Exercises This section will review the concepts and operations discussed in the previous sections of this manual. Manual Ply Creation Review Exercise In this section, manual plies will be created to define a composite part. The plies for the first half of the laminate will be created manually.

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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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