

Catia V6 Manual

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In CATIA V6 Part Design and Sketcher, you learned how to generate parts. The primary objective of this class is to utilize those parts to create a complex assembly of those parts that can be later used in stress analysis, kinematics, fitting simulations, and other areas.

~~TABLE OF CONTENTS~~

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The newest version, CATIA V6, came out in 2008 and has followed the same yearly version-history release pattern (and incremental improvements) as V5. However, it has been almost 10 years since V6 was released and engineers and designers everywhere are eagerly anticipating the presumed radical advancements that will be included in CATIA V7.

~~CATIA V7: When will it be released? — 3DSMAN~~

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