

# ADM 116: Introduction to CATIA

Introduction to parametric, three-dimensional modeling using CATIA (v5 or 6). Focus on how to navigate within this software, how to create three-dimensional solid models using industry best practices, and then how to create and manipulate assemblies made from these parts. Learn the process of designing models with CATIA from conceptual sketching, through to solid modeling, assembly design, and drawing production. Upon completion of this course you will have acquired the skills to confidently work with CATIA. Gain an understanding of the parametric design philosophy of CATIA in this extensive hands-on course. It is expected that all new users of CATIA will require this course.

**Credits:** 3

**Lab Hours:** 0

**Lecture Hours:** 3

**Prerequisites:**

None

**Co-Requisites:**

None