

ADM 116 : INTRODUCTION TO CATIA

Introduction to parametric, three-dimensional modeling using CATIA. The course focuses on how to navigate within the software, how to create three-dimensional solid models using industry best practices, and how to create and manipulate assemblies made from these parts. Students learn the process of designing models with CATIA from conceptual sketching, to solid modeling, assembly design, and drawing production. Upon completion of this course, users will have acquired the skills to confidently work with CATIA. Students will gain an understanding of the parametric design philosophy of CATIA in this extensive hands-on course.

Credits 3

Lab Hours 0

Theory Hours 3