CATIA

CATIA an acronym of computer-aided three-dimensional interactive application) is a multi-platform software suite for computer-aided design (CAD), computer-aided manufacturing (CAM), computer-aided engineering (CAE), PLM and 3D, developed by the French company Dassault Systèmes.

KEY FEATURES

Effective Upskilling Planned Curriculum
Team Learning Awesome Quizzes
Complete Hands on

The below Curriculum is Scheduled for 2 weeks

CURRICULUM

Module 1. INTRODUCTION
1.1 product realization process
1.2 brief history of cad/cam development
1.3 definition of cad/cam/cae
1.4 scope of this tutorial

Module 2. GETTING STARTED
2.1 starting nx9.0 session and opening files
2.2 printing, saving and closing part files
2.3 nx9.0 interface
2.4 coordinate systems
2.5 using layers
2.6 important commands/dialogs

Module 3. FORM FEATURES
3.1 overview
3.2 types of features
3.4 reference features
3.5 swept features
3.6 remove features 3.7 exercise - model a washer

Module 4. FEATURE OPERATIONS
4.1 overview
4.2 types of feature operations
4.3 feature operations on models
4.4 exercise - model a circular base

Module 5. DRAFTING
5.1 overview
5.2 drafting of models
5.3 exercise - drafting and dimensioning of a circular base

Module 6. SKETCHING
6.1 overview
6.2 sketching for creating models
6.3 exercises

Module 7. FREEFORM FEATURE
7.1 overview
7.2 freeform feature modeling
7.3 exercise - model a mouse

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CURRICULUM

Module 8. ASSEMBLY MODELING

8.1 overview

8.2 terminologies

8.3 assembly models

8.4 assembly navigator

8.5 mating conditions

8.6 impeller assembly

8.7 exploded view of impeller assembly

8.7 exercise - arbor press assembly