

Course Name : ***NX CAD (Sketching to Simulation)***
Course Duration : 40 Hrs.

Course overview

- **Intended audience**
 - This course is suited for designers, engineers, manufacturing engineers, application programmers, NC programmers, CAD/CAM managers, and system managers who need to manage and use NX.
- **Prerequisites**
 - **Education:** Diploma or UG 1st year completed in any one of following Streams.
 - Aeronautical, Automobile, Civil, Industrial, Marine, Mechanical, Mechatronics, Metallurgy, Production and Manufacturing Engineering.
 - **Software:** **None**
- **Course objectives**
 - After successfully completing this course, you should be able to:
 - Open and examine NX models.
 - Create and edit parametric solid models.
 - Create and modify basic assembly structures.
 - Create and modify simple drawings.
 - Combine rigid bodies (links), joints, and motion drivers to create a mechanism and set it into motion- Manage multiple motion simulations
- **Course Contents**
 - NX User interface
 - Create Basic Parts
 - Organize & Display Parts Model
 - Create Cylindrical Parts Using Sketches
 - Add Finishing Details
 - Simple Changes & Part Interrogation
 - Basic Parts Using Synchronous
 - Analyze Existing Assemblies
 - Bottom Up Assembly Building
 - Create a Basic Part Drawing
 - Links
 - Joint
 - Kinematics
 - Dynamics
 - Articulation
 - Couplers
 - Sequence
 - Arrangements